The Length Scale Effect between Wall and Closes Field Point on a Single Turbine Blade Simulation

Mohd Hafiz Mohd Noh, Ahmad Hussein Abdul Hamid
Faculty of Mechanical Engineering, Universiti Teknologi MARA, 40450 Selangor, Malaysia

Koichi Mori
Nagoya University, Japan

ABSTRACT

A numerical code has been developed based on experiment data, by using Finite Volume Method. Four different turbulence models (one equation model) have been tested; Spalart–Allmaras, Delay Eddy Simulation, Delay Detached Eddy Simulation and Improve Delayed Detached Eddy Simulation. The exit Reynolds number is 2.8×10^6 *and the exit isentropic Mach number is 0.79. The code has been validated with isentropic Mach number around the blade, trailing edge pressure distribution, vortex shedding frequency at S/D=-0.62 and boundary layer profile at a distance equal to one. DDES has shown a good agreement qualitatively and quantitatively with the experiment as compared with others and at region -1<S/D<1, all results show insignificant different.*

Keywords: *trailing edge pressure distribution, boundary layer, vortex shedding frequency, isentropic Mach number, one equation turbulence model*

Introduction

The research on turbine blade has been actively studied in line with the demand for better performance and efficiency. Since experimental verifications are costly and time consuming, hence more research has been directed towards computational simulations. The thorough challenge in

---

ISSN 1823- 5514, eISSN 2550-164X

Received for review: 2017-07-03
Accepted for publication: 2017-12-01
Published: 2018-05-15

turbine simulation is in validation part. Reyhani et. al. [1], has conducted a simulation using ANSYS CFX to simulate three-dimensional external flow and heat transfer simulation along with conduction analysis in solid domain. The results have been compared in terms of non-dimensional pressure and temperature, and the results have shown good agreement with experimental data. However some discrepancies have been detected on nominal axial from 0 to 0.9 A.K Saha et al. [2], have studied the flow effect on pressure side winglet and heat transfer over a blade tip. The simulation has been carried out using FLUENT. Although the validation is not presented on the trailing edge area, other than that, the results have shown good agreement with experiment in terms of pressure coefficient.

One of the important elements in higher Reynold number compressible flow numerical calculation is turbulent model. Aftab et. Al [3] have run six (6) different turbulent models in order to validate the pressure coefficient at six-degree angle of attack for two dimensional airfoil. The result shows γ-Re_θ SST turbulent model is the most accurate in captured the flow as the initial laminar separation. Under a high speed condition, vortex shedding normally occurred at the wake of turbine blade and this will decreased the base pressure [5]. In simulation, it required for finer grid in order to capture this shedding accurately especially near the wall, which also being proved experimentally by [6].

In almost most cases, the vortex shedding on a blade occured at wake and initialy occured at trailing edge. The effect of length scale calculation has been presented in this paper. A CFD numerical code has been developed based on the established experiment data from Von Karman Institute, Belgium. The calculation has been limited on one equation turbulent model only.

Nomenclature

C	:chord
C_{ax}	:axial chord
D	:trailing edge diameter
s	:pitch
P	:total pressure
T	:total temperature
M	:Mach number
S	:trailing edge length
L	:length of the hole
D_h	:hole's diameter
γ	:specific heat ratio (1.4)
f_d	:delay function (DDES)

v	:molecular viscosity
$U_{i,j,k}$	:velocity gradient in i, j and k direction
d_{DES}	:redefined DES length scale
C_{DES}	:adjustable parameter (0.65)
c	:speed of sound
c_0	:speed of sound in stagnation condition
v_T	:kinematic eddy viscosity
κ	:von Karman constant (0.41)
ξ	:wake loss
$\vec{V}$	:velocity vector
$\vec{n}$	:normal vector at inlet plane
V	:velocity
θ	:flow angle relative to the inlet boundary
c_0	:speed of sound at stagnation condition
$\Delta = \max(\Delta_x, \Delta_y, \Delta_z)$	:local maximum grid spacing

Subscripts

01	:inlet
02	:outlet
is	:isentropic
in	:inside the calculation domain
inlet	:inlet / boundary of the calculation domain

Experimental data (reference for validation)

Table 1: Blade configurations and flow conditions [6][7]

Configurations / Parametric	Title 2
Chord Length, C	140mm
Trailing Edge Diameter, D	7.43mm
Axial Chord Length, C_{ax}	91.84mm
Blade Pitch, s	97.44mm
Span-wise Length	14mm
Stagger Angle, $\lambda_{stagger}$	49.83deg
Inlet Total Pressure, P_{01}	140kPa
Inlet Total Temperature, T_{01}	280K

The experiment has been conducted by Sieverding et al. [6][7] and all the results are compared with this experiment data. The experimental conditions are illustrated in Table 1 and Figure 1. The measurement of the blade in this calculation is the same as the blade that used in the experiment, which has been summarized in Table 1. The only different is that the span-wise size in the calculation has been reduced, which become 10% from the actual span size in actual experiment in order to expedite calculation time. Table 1 described the overall flow conditions that used in this calculation and Figure 1 shows a blade that being used during experiment.

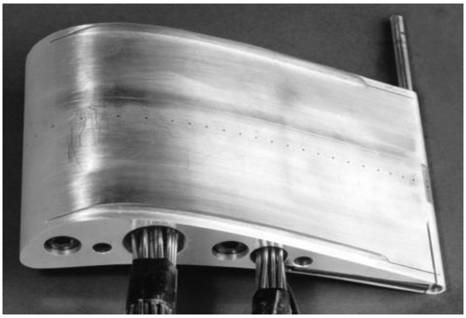

Figure 1: Actual blade during experiment [7].

Computational Method

The three-dimensional, Reynolds-averaged, unsteady compressible Navier–Stokes equation is solved. The inviscid fluxes are discretized using the total variation diminishing scheme, and viscous fluxes are discretized using standard central differences. The configuration of the cascade is presented in Figure 2, where all the parameter are similar with Table 1. The span-wise size of the blade is 10% of the chord length (14mm). *S* is the length along the blade trailing edge.

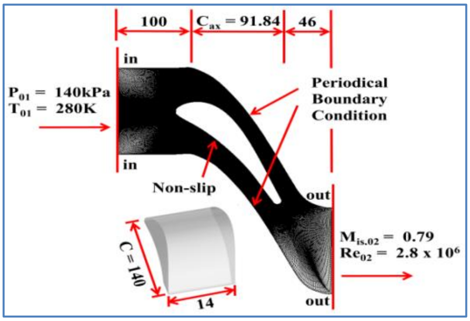

Figure 2: Simulation grid and boundary conditions

The code being used in this calculation is structured grid, single block, and 2nd order accurate in time. Roe's flux-difference splitting scheme is used for a numerical inviscid flux calculation, where 2nd order accuracy is achieved by the MUSCL scheme with the van Albada flux limiter. The unsteady flow is calculated by LUSGS together with a 2nd order dual time stepping method. The maximum number of iterations is 2 x 10^6, and there are five inner iterations for the implicit scheme (time marching). Dimensionless time step Δt^* is equal to 105.42×10^{-6} and the physical time step $\Delta t = 3.33 \times 10^{-7}$s with CFL value of 10. The numerical method is validated by comparing the numerical results with experimental data obtained by Sieverding et al., [6][7].

Boundary Condition

The isentropic flow calculation has been imposed at the inlet and outlet of the blade. Figure 2 shows an O-type structured grid that has 4.0×10^6 grid points. The pre-processing has been carried out by Gridgen Version 15 software. The minimum grid size is 0.002mm which is equivalent to y+ ≈ 1. On the wall, non-slip boundary condition is imposed as

$$u = v = w = 0 \ . \tag{1}$$

A total pressure of P_{01} = 140kPa and a total temperature, T_{01} = 280K, are applied at the inlet and subsonic Rieman Invarient condition, R_{01} has been imposed as inlet boundary condition [8].

$$R_{01} = \overrightarrow{V_{in}}\vec{n} - 2\left[\frac{c_{in}}{\gamma - 1}\right] \tag{2}$$

For inlet velocity V_{01}

$$V_{01} = \frac{-R_{01} - (\gamma - 1)}{(\gamma - 1)cos^2\theta + 2}\left\{1 + cos\theta\sqrt{A_1 - A_2}\right\} \tag{3}$$

where;

$$A_1 = \frac{[(\gamma - 1)cos^2\theta + 2]c_0^2}{(\gamma - 1)R_{01}^2} \quad and \quad A_2 = \left[\frac{\gamma - 1}{2}\right] \tag{4}$$

The c_0 can be calculated by;

$$c_0 = c_{in}^2 + \frac{\gamma - 1}{2}\left(\overrightarrow{V_{in}}\right)^2 \tag{5}$$

The computational domain consists of a single blade. Both sides of the domain (i.e. pessure side and suction side) are set as periodic boundary condition to prepresent a full-scale simulation (as shown in Figure 2). At the outlet, $M_{is,02}$ = 0.79 and a Reynolds number of 2.8 × 10^6, have been imposed [6][7]. The outlet static pressure is specified as P_{02} = 92,755Pa based on isentropic Mach number relation for $P_{s,02}$ and $T_{s,02}$;

$$P_{s,02} = P_{01}\left(1 + \frac{\gamma - 1}{2} M_{is,02}^2\right)^{-\left(\frac{\gamma}{\gamma-1}\right)} \tag{6}$$

$$T_{s,02} = T_{01}\left(1 + \frac{\gamma - 1}{2} M_{is,02}^2\right)^{-1} \tag{7}$$

Subsonic outlet boundary condition has been calculated by eq. (8),

$$R_{02} = \overrightarrow{V_{in}}\vec{n} + 2\left[\frac{c_{in}}{\gamma - 1}\right] \tag{8}$$

while the density, $\rho_{02}$and velocity V_{02} being calculated by;

$$\rho_{02} = \left[\frac{P_{02}}{P_{in}}\right]^{\left(\frac{1}{\gamma}\right)} \tag{9}$$

$$V_{02} = R_{02} - 2\left[\frac{c_{02}}{\gamma - 1}\right] \tag{10}$$

Turbulence Model

Since this study has been limited to one equation turbulent model, so Spalart-Allmaras turbulent model has been selected as basis for a transport equation model for eddy viscosity.

$$\frac{D\tilde{v}}{Dt} = \underbrace{c_{b1}\tilde{S}\tilde{v}}_{Production} + \underbrace{\frac{1}{\sigma}\left[\nabla.\left((v + \tilde{v})\nabla\tilde{v}\right) + c_{b2}(\nabla\tilde{v})^2\right]}_{Diffusion} - \underbrace{c_{w1}f_w\left(\frac{\tilde{v}}{d}\right)^2}_{Destruction} \tag{11}$$

Subscript *b* (on production and diffusion term) here stands for basic and "*d*" on destruction term is the distance from the wall to the nearest field point. $\tilde{S}$ is local deformation rate, $\tilde{v}$ is the working variable that satisfied the above equation [8]. On the destruction part, the length scale, "*d*" has been modified in three different approached. The first method works by replacing the length scale in the *SA* turbulence model d with a new length scale, $\tilde{d}$

$$\tilde{d} = min\left(d, (\max\left(\Delta_x, \Delta_y, \Delta_z\right))C_{DES}\right), \tag{12}$$

This approach is known as Delayed Eddy Simulation, *DES*. By doing this near the wall, the distance will be calculated as classic *d*, then beyond this point, the distance will be calculated as $\tilde{d} = \Delta C_{DES}$. Since the length scale only depends on the grid spacing, the challenge may occur if the grid spacing (in wall-normal) is smaller than boundary layer thickness [10]. Meaning that, the wall distance *d* is larger as compared with local maximum grid spacing Δ. In this case, the internal length scale is needed. To realize this, the parameter r_d is introduced as;

$$f_d = 1 - tanh([8r_d]^3) \quad where \;\; r_d = \frac{v_T + v}{\sqrt{u_{ijk}u_{ijk}}\kappa^2 d^2} \;\; , \tag{13}$$

f_d is equal to one (1) in the *LES* region where $r_d << 1$ and zero (0) are in the RANS region [10]. Finally, new length scale, d_{DES} can be obtained;

$$d_{DES} = d - f_d max(0, d - C_{DES}\Delta) \;. \tag{14}$$

This approached is known as Delay Detached Eddy Simulation, *DDES*. For *RANS* calculation, outside of the boundary layer and separated flow region, d_{DES} behaves like classical *DES*. In RANS simulation for boundary layer ($f_d = 0$), d_{DES} is calculated as a function of *d*, while beyond the boundary layer and separated flow region, d_{DES} is calculated as normal *DES*. A further improvement to *DDES* has been made by modifying the parameter r_d as presented in equation (15)

$$r_{d,iddes} = \frac{v_T + v}{\kappa^2 d^2 max\{[\sum_{ijk}\sqrt{u_{ijk}u_{ijk}}], 10^{-10}} \;\; . \tag{15}$$

The drawback of *DDES* and *IDDES* is majorly dedicated to higher requirement of computational time. On the other hand, both DDES and IDDES not necessary to develop very fine grid near the wall (of blade). This give significant improvement on *DES* part as the calculation not limited to grid size only.

Results and discussion

Initially, two dimensional calculations have been carried out in order to see the effect of three dimensionalities on turbine blade's analysis. The result has been tabulated on Figure 3.

As we can see on the figure, three dimensionalities has been a vital procedure need to be considered in order to analysis of turbine blade. The result shows significant discrepancies between experiment and simulation. The validation process for three dimensional calculations started with the isentropic Mach number around the blade for each turbulent model. This calculation has been done on the region between -1<S/D<+1, presented on Figure 4. We can see that changing the length scale on *S-A* model seems less effected for isentropic Mach number value. All the turbulent models almost agree well with the experiment. This is because there are no separation and vortex shedding initiate along this region. So modification on length scale doesn't give the influence on isentropic Mach number value around the blade, which also agreed by [6]. This situation is completely different at the area of trailing edge, as tabulated on Figure 5.

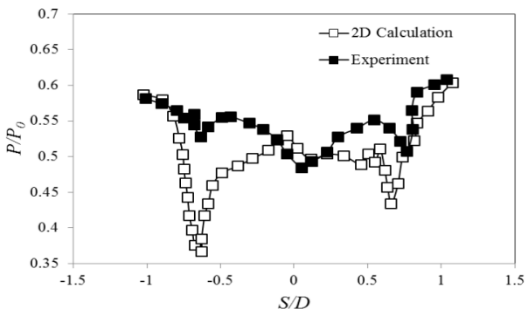

Figure 3 Trailing Edge Pressure Distribution comparisons

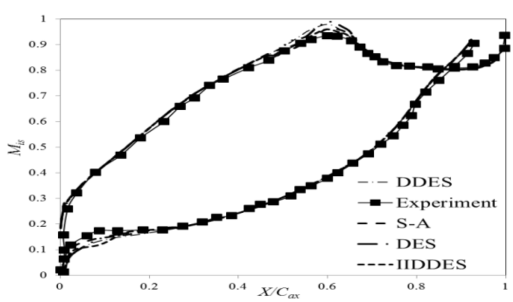

Figure 4 Isentropic Mach number around the blade

Figure 5 shows the result of different turbulence model on trailing edge pressure distribution. Vortex shedding in turbine blade occurred initially at S/D = ± 1 [12]. Obviously, *S-A* turbulent model is not enough to capture the minimal pressure that observed in the experiment on the region at *S/D* ≈ −0.7, 0.0 and 0.75. This is due to *d* on *S-A* model destruction part particularly depends on grid size, so if any viscous take place lower than this grid size, the effect may cannot be calculated.

This also being support by Potsdam et al., is that the standard *S-A* model is not suitable when dealing with high vortical flow calculation [16]. It can be concluded that *d* cannot be used in a conventional way. As for *DES* calculation, we can observed a slight improvement in capturing the pressure minima on the blade surface, but it is still far from the experiment result. Nevertheless, *DDES* and *IDDES* models has give good results, close to the experiment, which was also in-line with previous research [11][14][15]. Magagnato et al. [13] also claimed that *DDES* was capable in capturing the incipient separation in transitional flow prediction on a turbine blade.

The computation produced three minimum values of dimensionless pressure, which also similar as the experimental results (at $S/D \approx -0.7$, 0.0 and 0.75). The locations of computed result minimal points and values are closes with the experimental results. Sieverding et al. [6] concluded that right after the pressure minima, there are separations occurred mainly due to overexpansion on the pressure sides and the suction sides of the blade. This is believed to be one of the reasons why the argument between the present results and the experimental results at $|S/D| > 0.8$.

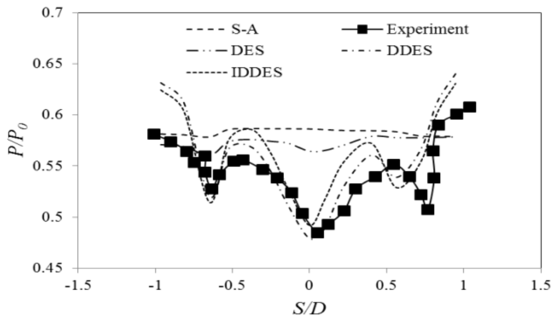

Figure 5 Trailing edge pressure distributions

The result also validated with the experiment data in term of vortex shedding frequency. In order to obtain this, Fast Fourier Transformation (FFT) calculation has been performed. FFT gives information about the dominant frequency at the chosen location and give indication whether the flow calculation has able to capture the calculation flow field in a timely accurate manner. FFT shows that the predominant frequency occurred at *S/D* = -0.62, as shown in Figure 6. Different turbulent models has outcome for different frequency peak values. The location of S/D = -0.62 has been chosen since the experiment data also being established at the same location. The first three frequency peaks results of computational and experimental are presented in Table 2.

S-A resulted largest different for all peaks, as compared with experiment results. *DDES* has given the closes result to the experiment compared others, where all peaks obtained agree well with each other, as in

Table 2. Besides that, the validation of the boundary layer characteristics also being calculated at a distance equal to one trailing edge diameter for the pressure side.

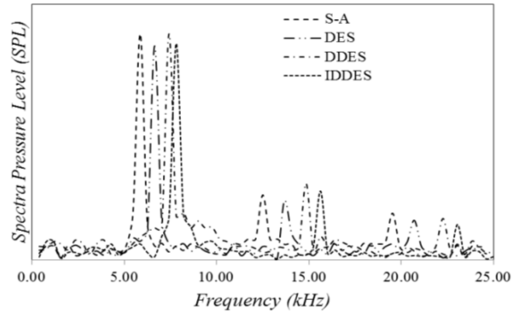

Figure 6 Shedding Frequency at S/D = -0.62.

Table 2: Comparison the value of frequency (kHz)

Peak	1st peak	Δ%	2nd peak	Δ%	3rd peak	Δ%
Experiment	7.37	-	14.71	-	22.14	-
S-A	5.90	19.95	12.50	15.02	19.53	11.79
DES	6.64	9.91	13.67	7.07	20.71	6.46
DDES	7.42	0.67	14.84	0.09	22.27	0.59
IDDES	7.82	6.11	15.62	6.19	23.06	4.16

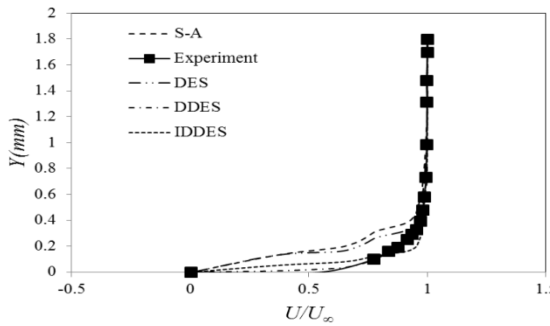

Figure 7 Boundary layer profiles at trailing edge (pressure side)

Figure 7 shows a good agreement has been achieved between experimental and computational results for *DDES*. Again, we can see *S-A* cannot captured velocity magnitude near the wall, as well as *DES*. *IDDES* has over estimated the captured u-velocity, esspecially at 0.13mm < y < 0.5mm region.

Conclusions

The effect of length scale from the wall to the closses point has been investigated. It was found that non-modified *S-A* turbulent model still relevant to be used for most all calculation, except at trailing edge area. This is useful in order to expedite the calculation time. Furthermore, *S-A DDES* turbulent model give the closes result to the experiment, as compared with others.

Acknowledgements

This research was supported by the LESTARI Research Grant (196/2017), provided by the Universiti Teknologi MARA (UiTM)

References

[1] Reyhani M.J., Alizadeth M., Fathi A., and Khaledi H., "Turbine Blade Temperature Calculation and Life Estimation – a Sensitivity Analysis", Elsivier B. V., National Library for Aeronautics and Astronautics, (2013).

[2] Saha A.K., Acharya S., and Bunker R., "Blade Tip Leakage Flow and Heat Transfer with Pressure Side Winglet", International Journal of Rotating Machinery, Volume 2006, Hindawi Publishing Corporation, pg 1-15 (2006).

[3] Aftab S.M.A., Rafie M.A.S., Razak N.A., and Kamal K.A., "Turbulence Model Selection for Low Reynolds Number Flow" PLOS ONE, DOI:10.1371/journal.pone.0153755 (2016).

[4] Cicatelli, G., and Serveiding, C.H., "The Effect of Vortex Shedding on the Unsteady Pressure Distribution around Trailing Edge of a Turbine Blade", Transaction of ASME, ASME Journal of Turbomachinery Vol.119, pp.810-819 (1997).

[5] Gostelow, J.P., Platzer, M.F., and Carscallen, W.E., "On Vortex Formation in the Wake Flows of Transonic Turbine Blades and Oscillating Airfoils", ASME Journal of Turbomachinery, Vol.128, pp.528-535 (2006).

[6] Sieverding, C.H., Ottolia, D., Bagner, C., Comadoro, A., Brouckaert, J.F., and Desse, J., "Unsteady Turbine Blade Wake

Characteristics", ASME Journal of Turbomachinery, Vol.126, pp.551-559 (2004).
[7] Sieverding, C.H., Richard, H. and Desse, J., "Turbine Blade Trailing Edge Flow Characteristics at High Subsonic Outlet Mach Number", ASME Journal of Turbomachinery, Vol.125, pp.298-309 (2003).
[8] Blazek, J., "Computational Fluid Dynamics; Principles and Applications", Elsevier, pp.267-293 (2001).
[9] Jeffrey, P.B., Sondergaard, R., and Rivir, R.B., "Turbine Separation Control Using Pulsed Vortex Generator Jets", Transaction of ASME, Journal of Turbomachinery, Vol.123, pp.198-206, (2001).
[10] Spalart, P.R., Jou,W.-H., Strelets, M., and Allmaras, S.R., "Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach", Advances in DNS/LES, pp.137-147 (1997).
[11] Spalart, P.R., Deck, S., Shur, M.L., Squires, K.D., Strelets, M.K. and Travin, A., "A New Version of Detach-Eddy Simulation, Resistant to ambiguous Grid Densities", Theoretical Computational Fluid Dynamics, Vol.20, pp.181-195 (2006).
[12] Lake, J.P., "Flow Separation Prevention on a Turbine Blade in Cascade at Low Reynolds Number", PhD Dissertation, Air Force Institute of Technology, AFIT/DS/ENY/99-01 (1999).
[13] Magagnato, F., and Gabi, M., "Prediction of Transitional Flow of Turbine Blades with DDES and LES", Proceedings of 8th World Congress on Computational Mechanics, July 2008 (2008).
[14] El-Gendi, Mohammed, K.I., Mori, K., and Nakamura, Y., "Novel Flow Control Method of Vortex Shedding for a Turbine Blade", Transaction Japan Society of Aeronautic Space Science, Vol.53, pp.122-129 (2010).
[15] M.El-Ghondour, Shatat, M.M.E., and Nakamura, Y., "On the physics of Vortex Formation at the Tip of a Turbine Blade", ASME Turbo Expo 2012: Turbine Technical Conference and Exposition, June 11-15 (2012).
[16] Potsdam, M., and Pullian, T., "Turbulence modeling treatment for rotorcraft wakes", In AHS Specialist's Conference on Aeromechanics, Jan. 23-25 (2008).