Investigations of active boundary layer control using DDES

Andreas Rumpf\textsuperscript{1,2}, Galih Bangga\textsuperscript{1}  
\textsuperscript{1}Institute of Aerodynamics and Gas Dynamics (IAG), University of Stuttgart, Pfaffenwaldring 21, 70569 Stuttgart, Germany  
\textsuperscript{2}University of Applied Sciences Esslingen, Faculty Building Services, Energy, Environment, Kanalstraße 33, 73728 Esslingen, Germany  
E-mail: Andreas.Rumpf@hs-esslingen.de and bangga@iag.uni-stuttgart.de  

Abstract. The root area of a wind turbine rotor is usually constructed by very thick airfoils with a relative thickness of more than 35%. An airfoil at that thickness is characterized with strong flow separation. To solve this issue, vortex generators can be used as an active control in order to stabilize the airflow and improve the aerodynamic performance, consequently. Boundary layer control however, investigated employing numerical techniques, strongly depend on the employed method as well as the mesh especially due to a weak vortex conservation in the RANS model. Using the method of Delayed detached eddy simulation (DDES) the solution can benefit from effort-efficient boundary layer modeling as well as LES vortex resolving apart from the surface. The present studies aims to investigate suitable numerical approach mesh dependencies in comparison to experimental data, for application to thick airfoil in future studies.

Keywords: active flow control, CFD, continuous jet, thick airfoil

1. Introduction

Aerodynamic profiles with a thickness of 40%, which become more present with increasing rotor diameter, suffer from detached flow. It has been demonstrated in \cite{1,2} that the maximum lift coefficient reduces significantly and the stall angle takes place at a very small angle of attack, indicating a bad behavior of the airfoil performance. This condition can lead to a reduced performance on the mechanical equipment employing those airfoil. It has been shown for example by Gbadebo et al. \cite{3}, Dring et al. \cite{4} and Dong et al. \cite{5} that 3D flow separation in the root area of a compressor can reduce the overall performance of the machine significantly. Several attempts have been done to delay flow separation by applying vortex generator jets (VGJs) \cite{6–9}. Many investigations of active boundary layer control consider a 2D flow, but meet the limit of interpretability due to 3D effects. Latest numerical approaches like delayed-detached eddy simulation (DDES) should give substantiated results in order to provide a three dimensional understanding of applied VGJs. For this purpose, systematic studies are presented in this work to assess the vortex characteristics generated by a single and pair of VGJs with several configurations employing different refined meshes in all directions along the vortex propagation. The studies shall be beneficial for the readers dealing with flow simulations to design and to prepare the appropriate high fidelity simulation setup, but also for wider range of engineers.
working in the field of wind energy.

2. Numerical setup for multiple span-wise distributed blowers

The impact of span wise distributed generated vortex structures on the boundary layer is
investigated in the following using different spatial discretisations at the expected vortex location.
Numerically obtained near-wall results are compared with experimental data from Johnston et
al [10]. Conclusions of the performed grid study are adopted to simulate the jet performance at
various yaw angle. The results are compared against available experimental data from Compton
et al. [11]. The setup, shown in Figure 1 consists of a flat plate, located along the green
marked area. The origin is chosen to be located downwind of the beginning of the flat plate

![Figure 1: Applied boundary conditions of the coarse background mesh for multiple span-wise distributed blowers](image1)

![Figure 2: (a) Visualization of the inclination angle $\beta_1$ for one respective blower $i$ - View in
positive $z$, (b) of the yaw angle $\beta_2$ for one respective blower $i$ - View in positive $y$ and (c) of
the applied boundary conditions for all blowers.](image2)

at $x/D = -85$ in order to reach the boundary layer thickness of 1.5 cm as it was determined
in experimental data by Johnston et al. [10] at the position of the blowing. The dimensions
of the domain, that are normalised by the blower diameter $D$ are depicted in Figure 3. The
required stream-wise length until the blower was approximated for the current setup using the
Prandtl’s 1/5 power law. Six spanwise blowers $i$ were installed in the experiments along the
$z$-axis. A definition of the angles is given in Figure 2a for $\beta_1$ and in Figure 2b for $\beta_2$. Wall
is modeled as a no-slip condition indicated as "NS-WALL" in Figure 2c. The mass flux of
the continuous jet is attributed to the boundary "ACTUATOR". The inclination $\beta_{1,i}$ of each
blower was set to $45^\circ$, whereby a turn around the $\eta_i$ -axis with $\beta_{2,i} = \pm 45$ allows three types
of configurations. Due to an assumed property of symmetry beside the $x − y$-plane, but also to
Figure 3: Dimensions of the computed domain in relation of the blower diameter $D$. (a) Span-wise extension $z/D = 32.124$ until $z/D = 0$. (b) View of the computed domain including the background meshes and the blowers - View along the $z$-Axis.

Figure 4: a) Integration of multiple blowers in the background meshes - View along the $x$-Axis in flow direction and b) detailed view of the overlapping area of adjacent meshes for the Chimera-Technique reduce computational effort, only two instead of all six blowers are considered for the numerical setup. By varying $\beta_{2,i}$, the blower can be set in a common direction, towards, or away from each other. The impact of the remaining four blowers in the experimental setup is simulated using the "PERIODIC" boundary conditions considering an endless amount of blowers in $z$-direction. The boundary condition is illustrated in Figure 1. An airflow is defined in positive $x$-direction with an airspeed of $15 \text{ m/s}$. The spatial dimensions of the setup are non-dimensioned with the blowing diameter $D = 6.35 \text{ mm}$ as depicted in Figure 3. Figure 3a depicts the dimensions from $z/D = -32.124$ to 0 where also the "PERIODIC" boundary conditions are applied. Much bigger is the expansion in $y$-direction as it also can be seen in Figure 3b. The actual blue highlighted region of interest (ROI) appears small compared to the green highlighted coarse background mesh. This circumstance guaranties homogeneous flow conditions around the ROI and the additional amount of cells is acceptable. The main part of the amount of cells contributes with 98% the blue highlighted fine background mesh together with the set of blowers as depicted in Figure 4a. The generated vortices are expected to display within the fine background mesh. The overlap of different adjacent meshes that is enabled by the chimera technique is shown in Figure 4b. An interpolation is carried out there using at least four cell layers. The blower mesh contains a mesh that represents the volume of the supplied air of the vortex generator. It is dimensioned lengthwise to 2.5 times of the blower diameter $D$ in order to provide a fully developed turbulent air flow. The computations were carried out using the hybrid RANS/LES approach employing the Menter SST [12] turbulence model within the boundary layer area. The Jameson-Schmidt-Turkel (JST) [13] approach was used for flux discretisation. This gives second order accuracy in space on smooth meshes. Dual time stepping according to Jameson [14] was applied for time.
accurate computations. The hybrid time integration was carried out using the explicit 5-stage Runge-Kutta central differencing scheme [13] for main equations and the Diagonal Dominant Alternating Direction Implicit (DDADI) scheme [15] in case of turbulence.

3. Flow field of a single blower on a flat plate at various yaw angle

3.1. Impacts of spatial discretisation on VGJ performances

In the following, aspect ratios of the equispaced fine background mesh was varied in the framework of a performed grid study. Bangga et al. [16] recently remarked an impact of different resolved meshes on the blade loads of a wind turbine. Further on, the wall normal density of cells that is dedicated to the representation of the boundary layer (BL) is varied in the following. An overview of the applied fine background meshes is given in Table 1. The number of grid cells ranging from 7,488,000 until more than 4 times is evaluated. A direct impact on the BL is expressed by the local, free-stream normalised skin friction coefficient $c_{f_{\infty}}$ on the investigated surface. At two selected downstream lines that refer to the experimental setup from Johnston et al. [10], the $c_{f_{\infty}}$ values are evaluated on the surface for the set of meshes in Table 1 and illustrated in Figure 5. The measurements are located at $x/D = 48.03$ and $x/D = 144.10$ and are named L2 and L4, respectively. Figure 5 firstly depicts experimental data that are marked with squares and linked with black dashed lines. The data correspond to Figure 4 in [10]. The simulated area extends from $z/D = -32.126$ to 0. The depicted simulated solution can be added from 0 to $z/D = 32.126$ due to the applied "PERIODIC" boundary condition in order to complete the range of the available experimental data. Maxima and minima of the dimensionless skin friction $c_{f_{\infty}}$ are shaped by vortex generator jets that are installed as blowers at the locations of $z/D = \pm 8.03$ and $z/D = \pm 24.10$. Taking Mesh A as a first reference to the experimental data, the characteristic evolution of $c_{f_{\infty}}$ over $z/D$ from CFD mostly over-

Table 1: Variations of the fine background mesh

| Mesh  | Amount of BL Cells | Height of BL $\Delta \frac{x}{D}$ | $\Delta \frac{y}{D}$ | $\Delta \frac{z}{D}$ | Amount of Cells |
|-------|--------------------|-----------------------------------|---------------------|---------------------|-----------------|
| Fine A | 32                 | 0.07                              | 0.47                | 0.16                | 1.7             | 11,648,000      |
| Fine B | 32                 | 0.07                              | 0.47                | 0.31                | 0.5             | 7,488,000       |
| Fine C | 64                 | 1.57                              | 0.47                | 0.27                | 0.31            | 0.85            | 9,152,000       |
| Fine D | 32                 | 0.07                              | 0.24                | 0.31                | 0.31            | 1               | 11,648,000      |
| Fine E | 64                 | 1.57                              | 0.47                | 0.39                | 0.16            | 2.5             | 16,640,000      |
| Fine F | 64                 | 1.57                              | 0.47                | 0.39                | 0.08            | 5               | 32,640,000      |

Figure 5: Span-wise free stream normalised wall friction $c_{f_{\infty}}$ at two different stream-wise locations after the VGJ creation using different refined meshes.
predicts the experiment with a constant value of $\Delta c_f = 2e^{-4}$. Decreasing the aspect ratio $\Delta x/\Delta y$ from 1.7 over 1 to 0.5 leads to smaller extreme values, but also the higher gradient of $c_f$ wanders away from the grey marked position of the blower. Only meshes with an aspect ratio $\Delta x/\Delta y > 1$ correlate with the homogeneously appearing values within the range from $z/D > 20$ and $z/D < 12$. The importance of the aspect ratio is underlined comparing Mesh A with Mesh D. Both meshes include the same amount of cells, but the evolution of $c_f$ strongly differs. The reduction of the amount of the extreme values can be realized by increasing the amount of boundary layer resolving cells with a given growth rate. The increase from 32 to 64 cells generally leads to under-predicted values of the skin friction. By this way, the fine spaced boundary layer cells create a thicker layer and equally cause a higher aspect ratio. The total height of the layer in each mesh is shown in Table 1. Finally, it can be concluded that the aspect ratios between 1 and 5 are suited to match local skin friction that is influenced by vortex generator jets. Comparing Figure 5a at L2 with Figure 5b at L4 it can be noticed, that the decrease downstream and a weakening of the peaks can be reproduced. Furthermore, the experimental data show a shift of the distinct peaks turn around $z/D = -6$ at L2 to $z/D = -4$ at L4.

3.2. Streamwise vortex propagation

The vortex cores within the domain are detected using eigenvalues and eigenvectors of the velocity gradient tensors employing a post-processing tool Tecplot. The coordinates of the detected vortex cores are depicted in Figure 6 as symbols that are linked with a dashed line for each corresponding mesh. The created vortices are shown in two different views ($x - y$ and $x - z$) to enable the comparison of the spanwise and wall normal propagation. Considering the $x - y$-view, the obtained values appear to be characterized by three distinct properties. The almost collapsing values from mesh E and mesh F have a comparative high value of aspect ratio $\Delta x/\Delta y$ of 2.5 and 5, the highest applied amount of cells in total as well as a comparative fine resolution in the region of the boundary layer. The vortices mentioned show up the highest spanwise propagation of $\Delta z/D = 2.8$ units away from the their origin at the respective blower. A particular low value of $\Delta z/D = 1.8$ applies to the vortices in mesh D, with an exclusive fine discretisation in $x/D$-direction. Its aspect ratio of 1 can not be considered as the decisive factor leading to the lowest amount of $\Delta z/D$. The vortex propagation that lies between the enumerated vortices have comparative moderate aspect ratios from 0.5 to 1.7. However, the values of mesh C match the interval of values that refer to mesh E and F after $x/D = 100$. Hence, mesh C, E and F that include 64 cells in the region of the boundary layer reveal a
similar spanwise vortex propagation after \( x/D = 100 \). The vortices in mesh D radiate furthest from their points of origin. The linear like wall normal propagation of vortices in all meshes significantly differs from the qualitative progression in \( z/D \)-direction. The steadily growing wall-normal distance with increasing \( x/D \) allows vortices to extend across the boundary layer and out into the free-stream as it was observed in [10]. The progression of the vortices in \( z/D \)-direction correlates with an inverted exponential function \( (z/D) = 1/(x/D)^n \), with \( n > 0 \). With respect to the "PERIODIC" boundary condition, duplicates of the present data can be added every \( z/D = 32.124 \), which comes from the span-wise extension of the domain. Note that the geometry spans from \( z/D = -32.124 \) to \( z/D = 0 \), as it can be seen in Figure 4a. Consequently, the vortex located on opposite sides of plane at \( z/D = 32.124 \) moves closer to each other from upstream at L2 to downstream at L4 as it was observed in the experimental setup in [10]. However, their convergence seems to be limited like the estimated functional connection \( (z/D) = 1/(x/D)^n \). This suggests a mutual induction of more closely spaced vortex pairs, dominating effect according to observations of Pauley et al. [17].

3.3. Mesh dependent vortex shape and vorticity

The preceded observations are based on the impact of the boundary cell layer height and aspect ratio. In the following, these two parameters are compared with the respective resulting vorticity in the \( y-z \) plane at L2. Figure 7 shows the composition consisting of the finer background mesh with its boundary layer cells. The applied amount of boundary layer cells, as they are given in Table 1, result in a respective summed height of BL cells. The grid study contains a common height of the first cell of \( 2.24 \times 10^{-5} \) m on the boundary and a growth rate of 1.1. Depending on the amount of boundary cells, either 32 or 64, the summed cell height is \( 0.07 y/D \) or rather \( 1.57 \) times the blower diameter. Figure 7 shows a large horizontal expansion of the vorticity for the mesh F compared to the mesh that is build up by 32 BL cells. The vortex distribution of the remaining meshes was observed with a similar result. Regarding the vortex core which is located close to \( y/D = 2 \), the layer of BL cells covers with an amount of 64 cells at \( 1.57 y/D \) almost 80% of the lower half of the vortex. Due to wall normal mesh refinement, the local aspect ratio is approximately 1, provided that the mesh likewise is refined in \( z/D \)-direction. By the mathematical description of a Rankine vortex, the highest occurring vorticity is located at the vortex core. However, the present distribution of the vorticity is rectified due do the impact of the wall. The wall prevents the free symmetrical extension of the vortex leading to a stronger change of direction and a higher vorticity.

The vorticity decreases from 0.3 to a tenth of it further in wall normal direction where the high density of pathlines border the vortex. The horizontal expansion for aspect ratios of less than

Figure 7: Visualisation of the vorticity distribution at 48.03 blower diameter after the VGJ normalised as \( \omega_x = \Omega_x/\sqrt{\rho T_\infty} \). (a) Mesh D, (b) Mesh F.
equal one in Figure 7a is higher compared to the highest observed vertical expansion in Figure 7b. Mesh D shows the lowest value of vorticity of all compared vortices. Its stream-wise spatial discretisation is half sized, or rather twice as fine compared to the other meshes. Regarding the investigation performed by Küpper et al. [18], employing low aspect ratio, also a flattening of the cross stream vortex area was observed compared to experimental data from Bray et al. [19].

3.4. Impacts on the boundary layer

The numerical simulation facilitates the decomposition of the velocity vector. In that way, the impacts of stream-wise vortex on the boundary layer can be indicated due to its cross stream induced component. These phenomena are compared using different refined meshes in Figure 8. Up to a wall normal distance of 4.7 D, its subfigures illustrate the free stream normalised velocity profiles at four selected locations. The first mesh dependent velocity profiles are extracted at z/D = 25.04 where the experimental data in Figure 5a show a maximum of the c_{f\infty} path. The experimental data exhibit a symmetry around its center at z/D = 0, but with deviations. For instance, the peak at z/D = −8 corresponds to the peak at z/D = +9 with a deviation of one blower diameter. Nevertheless, the simulated data match the experimental location of one peak at z/D = 9 using mesh A, E and F. The velocity profiles in Figure 8a are orientated towards the experimentally given location at z/D = 25.04. This non-matched location, mesh A, E and F show up the closest approach to the experimental data regarding c_{f\infty} and z/D. This applies equally for the velocity profiles up to a wall normal height of y/D = 1. The subsequent drop in velocity can not be reproduced by any mesh. However, mesh A, E and F show again the best approach for y/D > 3. In general, the closest approach is sortable by the aspect ratio ∆y/∆z to the sequence: Mesh F(5), mesh E(2.5), mesh A(1.7), mesh C(0.85), mesh B(0.5) and mesh D(1). The latter mesh is characterised by a twice as fine discretisation in z/D. Regarding the normalised cross stream velocity at the range around y/D = 1.57, where the experimental data procast a local minima, mesh A, E and F have the highest proportion of the cross stream velocity. The enhanced cross stream velocity results in a lower velocity in flow direction. Consequently, also within the range of the local minima, mesh E and F represent the most physical solutions.

The following mesh dependent velocity profiles are extracted at z/D = 28.19, where the c_{f\infty}-path is the smallest. At this minima the z/D location of all c_{f\infty}-paths collapse with the experimental data. Moreover, meshes E and F match the c_{f\infty}-value. The proximity to the vortex is indicated by a comparably high cross stream component of up to 4.2% of the free-stream velocity.
This percentage is comparable to the other side of the vortex where the flow is moved towards the wall. Up to $y/D = 1$, mesh C shows the closest approach, but closely followed by mesh E and F. Around $y/D = 1.5$ the curve of the experimental data is slightly inflected that is reproduced most using mesh A. However, all simulated curves reproduce lower in flow velocities for the range up to $y/D < 1.5$ compared to the profiles at other locations in Figures 8a, 8c and 8d. The profiles in Figures 8c and 8d are built up with data at $z/D = 15.59$ and 0. They represent the wall referring point between two vortices where the flow is moved downwards, or rather upwards. The effect of the cross-flow in the vortex field cancels itself out to a percentage of less than 1% of the free stream. The cross stream percentage close to 0 at $z/D = 15.59$ in Figure 8c indicates a symmetrical flow field. At both latter positions the experimental data are matched using mesh E and F up to $z/D = 1.5$. However, the remaining meshes are closer concerning the prediction of the boundary layer thickness of $2y/D$. Regarding the boundary layer thickness for all selected locations, a local cross-stream refinement of the mesh is appropriate near the vortices where also the cross-stream velocity shows elevated levels. In contrast, the cross stream refinement of the grid discretisation leads to a thicker predicted boundary layer at locations that are away from vortices.

3.5. Downstream vortex observation of a single blower

A sufficient impact of a VG is expected along a surface. The state of the vortex at a more distant location 0.97m after its generation is given in Figure 9. The distance is chosen in accordance with the experimental investigation of Compton et al. [11]. As it can be seen in Figure 9a, the dashed marked numerical solution matches well the span-wise position of the experimental result, indicated by solid lines. Nevertheless, the wall-normal simulated expansion is approximately double of the reference. However, the rage of the vorticity values is well matched. Further on, the boundary layer can follow the experimentally detected shape in Figure 9b. With focus on the cross stream flow, also vectors indicating the turning vortex collapse with the data of Compton et al. [11] in Figure 9c.

3.6. Stream-wise velocity field with different yaw angles

The creation of vortices at different yaw angles is indicated in Figure 10. The wall-normal propagation in Figure 10b and the span-wise location along $x$ is detected evaluating eigenvalues of the local velocity gradient tensor. The wall normal propagation for the in-flow configuration ($\beta_2 = 0^\circ$) shows the vortex cores to be the closest to the wall compared to the other adjusted angles. Along the flow in $x$-direction, gaps of the vortex propagation are visible at all configurations. The location determinant algorithm shows up difficulties at these locations.
indicate how strong the vortices are pronounced. Clearly distinguishable are the cases, when the blower has no velocity component in spanwise direction regarding the symmetrical flow field that is marked by path-lines in the Figures 11a and 11e for $\beta_2 = 0^\circ$ and $\beta_2 = 180^\circ$, respectively. A weak distribution of the vorticity at around $\pm 100 \ [1/s]$ in Figure 11e compared to Figure 11a underlines the observation of weak counter-rotating vortex pairs in Figure 10. Counter-rotating vortices are characterised by dashed lines around a negative contour level as it can be seen in Figure 11a. Moreover the presence of a counter rotation vortex pair, secondary induced vortices can be observed rudimentary in Figure 11e. The amount of the vorticity is qualitatively higher in case of a present in-flow velocity component of the blower. The highest amount and extension of the vorticity level of over 500 $[1/s]$ is created by the blower with a pure spanwise velocity vector as it can be seen for $\beta_2 = 90^\circ$ in Figure 11c.

4. Conclusions

A grid study was performed and showed, that with an increasing span-wise mesh refinement, the span-wise vortex propagation increases as well. Over an span-wise to wall-normal spacing ratio of greater than 2.5, the vortex propagation collapses for different refinements. The cell size in flow-direction showed good agreement with experimental data on Johnston et al [10], if is defined higher than the average wall-normal dimension. An additional refinement in flow direction results in a significant dissipation of the vortex. Furthermore, the cross-flow spacing ratio influences the shape of the cross area of the vortex. Aspect ratios of smaller than one lead to a flattened extension as it also was observed in the simulations conducted by Küpper et al. [18]. Selected velocity profiles as well as the local free-stream normalised skin friction
showed the best agreement using mesh F of the present study with an aspect ratio of 5. The latter named mesh was employed in a simulation that corresponds the experiment conducted by Compton et al [11]. The stream-wise observed vorticity and the location of its highest amount were detected firstly twice as far away after 0.66 [m] of its generation. However, the vortex simulated vortex cores collapse with the experimental data after the second of four observed stream-wise locations. An increased dissipation of the vorticity may be observed comparing with the experiment. The impact on the boundary layer shows a good agreement, considering the stream-wise velocity component. The examination of different yaw angles $\beta_2$ showed a strong pronounced and stream-wise persistent vortex structure for the configuration $\beta_1 = 45^\circ$ and $\beta_1 = 90^\circ$. The vortex structure and its impact on the boundary layer for $\beta_2 = 45^\circ$ and $\beta_2 = 135^\circ$ are comparable. Blowing configurations in-flow-, or against-the-flow direction result in the formation of two counter-rotation vortex structures that dissipate soon compared to other yaw angles.

References

[1] JP Baker, EA Mayda, and CP Van Dam. Experimental analysis of thick blunt trailing-edge wind turbine airfoils. *Journal of Solar Energy Engineering*, 128(4):422–431, 2006.

[2] Galih Bangga, Thorsten Lutz, and Ewald Krämer. Numerical investigation of unsteady aerodynamic effects on thick flatback airfoils. *arXiv preprint arXiv:1705.04211*, 2017.

[3] Semiu A Gbadede, Nicholas A Cumpsty, and Tom P Hynes. Three-dimensional separations in axial compressors. *Journal of turbomachinery*, 127(2):331–339, 2005.

[4] RP Dring, HD Joslyn, and LW Hardin. An investigation of axial compressor rotor aerodynamics. *Journal of Engineering for Power*, 104(1):84–96, 1982.

[5] Y Dong, SJ Gallimore, and HP Hodson. Three-dimensional flows and loss reduction in axial compressors. *Journal of turbomachinery*, 109(3):354–361, 1987.

[6] Barton L Smith and Ari Glezer. The formation and evolution of synthetic jets. *Physics of Fluids (1994-present)*, 10(9):2281–2297, 1998.

[7] RA Wallis. The use of air jets for boundary layer control. Technical report, Aeronautical research labs Melbourne (Australia), 1952.

[8] Galih Bangga, Thorsten Lutz, and Ewald Krämer. Active separation control on a very thick wind turbine airfoil—a urans and ddes perspective. In *Journal of Physics: Conference Series*, volume 1037, page 022025. IOP Publishing, 2018.

[9] Prabhu Kankatala and Galih Bangga. Active separation control on thick wind turbine airfoils by means of steady and unsteady blowing. *Advanced Theory and Simulations*, 2(7):1900077, 2019.

[10] James P Johnston and Michihiro Nishi. Vortex generator jets-means for flow separation control. *AIAA journal*, 28(6):989–994, 1990.

[11] Debora A. Compton and James P. Johnston. Streamwise vortex production by pitched and skewed jets in a turbulent boundary layer. -1, 12 1990.

[12] Florian R Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, 32(8):1598–1605, 1994.

[13] Antony Jameson, Wolfgang Schmidt, and Eli Turkel. Numerical solution of the euler equations by finite volume methods using runge kutta time stepping schemes. In *14th fluid and plasma dynamics conference*, page 1259, 1981.

[14] Antony Jameson. Multigrid algorithms for compressible flow calculations. In *Multigrid Methods II*, pages 166–201. Springer, 1986.

[15] Jens K Fassbender. *Improved robustness for numerical simulation of turbulent flows around civil transport aircraft at flight Reynolds numbers*. PhD thesis, 2003.

[16] Galih Bangga, Pascal Weihing, Thorsten Lutz, and Ewald Kraemer. Effect of computational grid on accurate prediction of a wind turbine rotor using delayed detached-eddy simulations. *Journal of Mechanical Science and Technology*, 31(5):2359–2364, 2017.

[17] Wayne R Pauley and John K Eaton. Experimental study of the development of longitudinal vortex pairs embedded in a turbulent boundary layer. *AIAA journal*, 26(7):816–823, 1988.

[18] Christoph Kuepper and Frank S Henry. Numerical study of air-jet vortex generators in a turbulent boundary layer. *Applied Mathematical Modelling*, 27(5):359–377, 2003.

[19] Tim P Bray. A parametric study of vane and air-jet vortex generators. 1998.