The 8th International Conference on Jets, Wakes and Separated Flows, ICJWSF-2024
September 23-25, 2024, Firenze, Italy

Numerical prediction of the heat loads on a tubine vane test case: assessment of RANS approach capabilities

N. Castelli[1*], T. Bacci[1], A. Andreini[1] and S. Giuntini[2]

1: Dept. of Industrial Engineering, University of Florence, Florence, Italy
2: Baker Hughes - Nuovo Pignone Tecnologie s.r.l., Florence, Italy

[*] Corresponding author: niccolo.castelli@unifi.it

Abstract

Achieving a reliable prediction of heat transfer is crucial to optimize blade cooling and overall engine performance. This paper focuses on the assessment of the capability of RANS (Reynolds-Averaged Navier-Stokes) approaches to characterize the external loads on the LS-89 profile. To this end, experimental results of turbulence and heat transfer coefficients (HTC) have been be compared with numerical simulations using both RANS and LES (Large Eddy Simulation) approaches. Particular attention was also put in selecting the appropriate type of boundary condition at the domain inlet, especially regarding turbulence. This allowed to better understand the limitations of the turbulence models. As a crucial aspect related to heat loads prediction capabilities, the reliability of different transition models was also investigated.

Keyword: *Turbulence, Transition, Heat transfer, Boundary conditions*

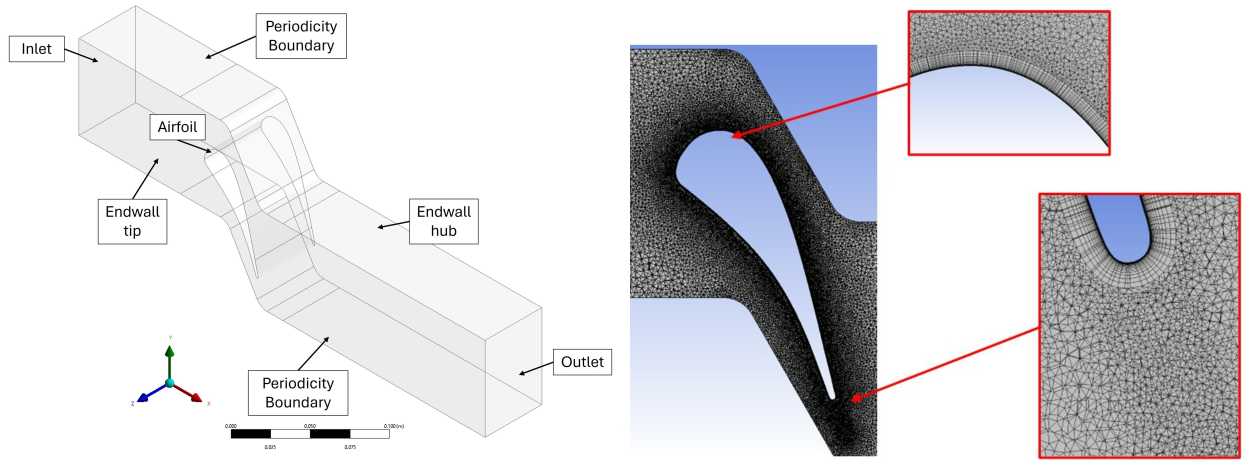
1. Introduction

The prediction of heat transfer distribution over turbine blade surfaces plays a major role in determining the heat load that the airfoil must withstand. Many efforts have been already done during the past decades by both academia and industry to establish the best numerical tools able to provide the most reliable estimations according to available experimental data. While several studies have pointed out to the necessity to adopt scale resolving approaches, especially when highly turbulent flows are concerned, the RANS tools, and their reliability, is still a topic of interest due to their reduced calculation time. In this regard, the VKI LS-89 profile is a relevant test case, as demonstrated by several numerical studies carried out over the years [1, 2, 3].

Moreover, a key feature to correctly evaluate the heat transfer around airfoils concerns the modeling of free-stream turbulence decay upstream the leading edge of the blade. Gourdain et al. [1] pointed out the role of a correct level of the turbulence intensity and length scale at the inlet in order to mimic the turbulence effects. For this reason, many studies focused on how replicate a grid generated turbulence in a numerical environment. Torrano et al. [4] evaluated the capability of different RANS turbulence models to reproduce the decay downstream a grid. [4, 5], mainly indicating the necessity to manipulate inlet boundary values to get the correct decay. The aim of the present work is to evaluate the capabilities of turbulence and transition RANS models to predict the heat load on a turbine vane. Attention to the impact of different strategies adopted to set the inlet turbulence condition was also investigated, as a strictly related aspect.

2. Test case

The 2D vane profile analyzed through both experiments and simulations is widely spread in the literature and it was developed by VKI [6]. In the present work, the original profile was magnified by a 1.3 scale, due to consistency with the airfoil employed during the experimental campaign. The experimental apparatus and measurement technique, as well as the detailed geometric parameters of the airfoil, can be found in [7].



(a) (b)

Fig. 1: (a) Three-dimensional computational domain of the VKI LS-89 test case and boundary conditions and (b) computational mesh of the periodic domain of the LS-89 vane, featuring a global volume sizing of $3mm$, surface sizing of $0.3mm$ (PS) and $0.25mm$ (SS), and 27 prismatic layers.

3. Computational domain

Achieving full fidelity with the experimental setup would need to employ a fluid domain that closely replicates the configuration of the test rig. This approach ensures that the simulation accurately reflects the conditions present in the actual experimental environment. However, in the case of simulations involving airfoil vanes, it is common practice replicates a single vane passage assuming periodicity in the tangential direction. This choice allows to simplify the numerical setup and to reduce the overall computational costs and it is justified by the capabilities of the test rig to replicate periodic conditions, shown in the test campaign.

The current domain is shown in Fig. 1a together with the boundary conditions. The inlet was positioned at approximately $x/C_{ax} = -2.3$, at the upstream position where turbulence intensity was measured using hot-wire anemometry technique, while the outlet of the domain is positioned sufficiently downstream of the trailing edge of the vane to ensure pressure uniformity at the discharge.

A hybrid unstructured grid was generated combining tetrahedrons in the mainstream and prismatic elements along the walls to allow a proper boundary layer treatment. At the end of a dedicated mesh sensitivity study, the global sizing in the fluid domain was set to $3mm$, while a surface sizing of $0.3mm$ on the pressure side and $0.25mm$ on the suction side was employed on the blade surface to ensure a high resolution near the wall. Regarding the boundary layer, a maximum thickness of about $1mm$ has been sought by imposing a minimum thickness of $0.5\mu m$ at the wall to ensure a y^+ lower than one everywhere on the airfoil, while a total number of 27 elements across the boundary layer has been selected to avoid a huge increase of the total number of elements. Fig. 1b shows some details of the generated mesh around the airfoil: the trailing edge region has been refined with a dedicated BoI in order to improve the solution in such a delicate zone. As a result, a 17.4 million elements grid was generated.

4. RANS numerical setup and boundary conditions

The whole numerical activity was conducted in the ANSYS Fluent 2023 R2 suite. The compressible Navier-Stokes equations were solved assuming the ideal gas law for the equation of state. Air specific heat at constant pressure, thermal conductivity and dynamic viscosity are given as fourth-grade temperature-dependent polynomial. Momentum and pressure-based continuity equations were solved simultaneously using the coupled algorithm to ensure a more robust convergence. Additionally, second-order discretization was applied to each variable.

The $k-\omega$ SST turbulence model [8] was used to resolve the turbulent term of the Reynolds stress tensor. This model accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. For the transition two different models implemented in Ansys Fluent have been were selected and a brief description is provided below:

- $\gamma - Re_\theta$ *Transition Model*: also referred as Transition SST Model, it is a four equation model based on the coupling of the $k - \omega$ SST transport equations with two other transport equations: one for the intermittency γ and one for the transition onset criteria in terms of momentum thickness Reynolds number Re_θ [9]. It utilizes an empirical correlation designed to account for both standard bypass transition and flows in low free-stream turbulence conditions which has been thoroughly validated alongside the SST turbulence model across a broad spectrum of transitional flows.

- γ *Transition Model*: also known as Intermittency Model, it is a further development of the previous model as it solves only one additional transport equation for the turbulence intermittency γ and avoids the need for the second equation for the transition onset criteria Re_θ [10]. This model has been fine-tuned against many turbomachinery and external aerodynamics test cases. Compared to the previous model, it offers the advantage of requiring fewer computational resources and the ability to capture cross-flow instabilities.

The measured and simulated test point was characterized by exit Mach and Reynolds number equal to 0.7 and 10^6 respectively. Both a high (15%) and low (1%) inlet turbulence condition was investigated. Turbulence values are defined as the ones at $x/C_{ax} = -1.5$ (1.5 chords upstream the vane LE).
Boundaries were imposed to match the experimentally measured mass flow rate and total-to-static pressure ratio across the vane. Regarding the turbulence at the inlet of the domain, it is important to distinguish between simulations conducted at low and high turbulence. In the first case, the numerical setup aims to replicate an experimental condition where the flow develops freely in the duct without any device to control the inlet turbulence level. In the second case, the simulation aims to replicate an experimental setup where the flow passes through a specifically designed grid to generate a specific turbulence level; two measurement positions were used during the test campaign ($x/C_{ax} = -2.3$ and $x/C_{ax} = -1.5$) to retrieve its decay. As already stated, special attention was given to reproduce the turbulence decay upstream of the vane by setting the inlet conditions properly, in order to evaluate its effect on the HTC distribution over the blade surface. In particular, different values of inlet turbulence level Tu_0 and the eddy length scale value l_0 will be imposed at the domain inlet.

Concerning the wall treatment, on both the hub and tip endwall of the computational domain, a no-slip condition and adiabatic thermal boundary condition were applied. A no-slip condition was also imposed on the airfoil surface, while regarding boundary conditions for thermal characterization a steady-state approach was selected. This requires two distinct simulations having different thermal boundary condition on airfoil wall: an adiabatic wall simulation (to retrieve the adiabatic wall temperature) and a second one with imposed wall temperature, from which the heat flux is calculated. The heat transfer coefficient (HTC) can be then retrieved from the latter and the known temperature difference (calculated adiabatic wall minus imposed wall temperature). The HTC along the abscissa of the airfoil was then spanwise averaged along 60% of the blade span, to produce 1D profiles.

5. Scale-resolving modeling

Concerning the LES calculations, a scale-resolved method is mandatory to precisely capture the vortex shedding of unsteady flows and their impact on the vane's boundary layers. When handling turbulent flows, LES is the most suitable method because it does not rely on turbulence models since the subgrid-scale (SGS) model has only a minor influence on the turbulence mixing, allowing for more accurate representation of flow dynamics. Even if a wall-resolved LES is very expensive because of the cost associated with the resolution of the near-wall region, it is believed to be the most effective approach. The WALE model from Nicoud and Ducros [11] was selected to provide a closure to the subgrid-scale turbulence since it is known to behave well in the near-wall region and with transitional flows. Compared to RANS setup, several changes were implemented to ensure the right generation and transport of turbulent eddies. First, a velocity boundary condition was applied at the inlet in order to use the Synthetic Turbulence Generator by Shur et al. [12] to create fluctuations at the domain entrance, while turbulence intensity and eddy length scale were set as done for the RANS, in particular $Tu_0 = 18\%$ and $l_0 = 12mm$. Additionally, the mesh was refined both globally and locally. The overall mesh size was decreased from $3.0mm$ to $1.5mm$, while a BoI with a refinement of $0.4mm$ at the vane endwalls enabled finer discretization for the secondary flows developing in this region. Surface sizing on pressure side and suction side of the blade was turned down to $0.15mm$ and $0.1mm$, respectively. It is worth reminding the current sizing are a trade-off between computational cost and spatial discretization to ensure reasonable values of x^+ and z^+ and hence, to control the resolution in all three space dimensions according to the wall-resolved LES requirements. As a result, a mesh with 135 million elements was achieved. The mesh quality was assessed by means of Celik index [13] which should take values greater than 0.75 to guarantee a sufficiently resolved turbulent flow solution. For the current calculation, the Celik index was found equal to 0.9 or higher everywhere in the computational domain.

6. Results and discussion

In this section, the heat transfer coefficient prediction coming from all the simulations will be compared with experimental values highlighting the role of an appropriate turbulence modelling for RANS calculations.

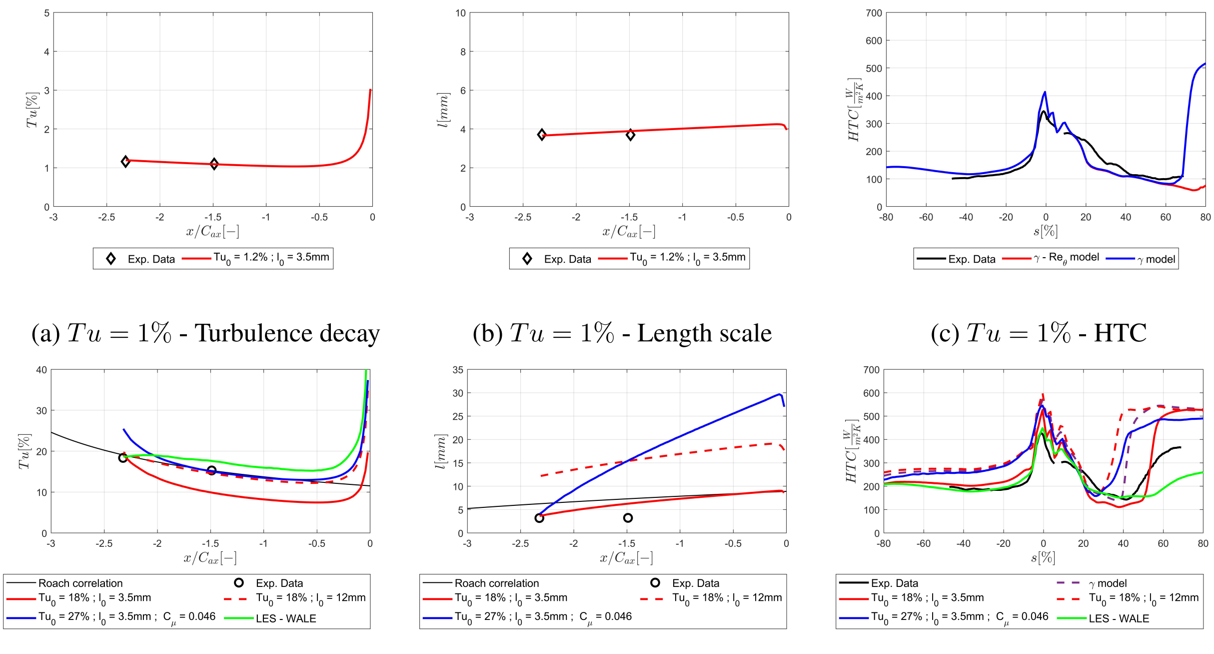
The choice of boundary conditions for turbulence at the inlet becomes crucial to reproduce the measured turbulence decay from the domain inlet to the leading edge of the blade. Turbulence intensity and length scale downstream the domain inlet were computed according to the following equations:

$$Tu = \frac{\sqrt{\frac{2}{3}k}}{|v|} \qquad l = \frac{\sqrt{k}}{C_\mu \cdot \omega} \tag{1}$$

where $|v|$ is the velocity magnitude, C_μ is a $k-\omega$ model calibration constant (0.09 default value), ω is the eddies specific dissipation rate and k is the turbulent kinetic energy which for the LES calculation, has been evaluated from the fluctuations of the velocity components.

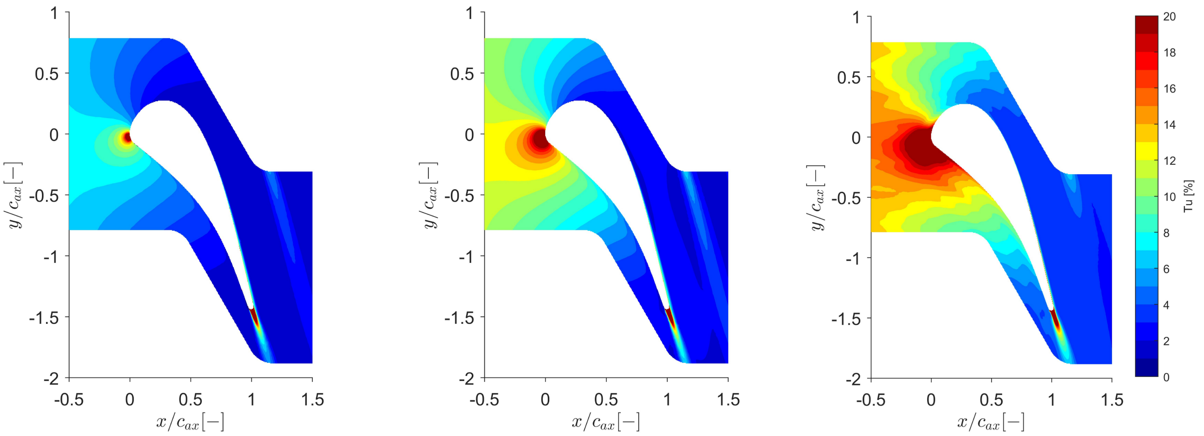
As shown by Fig. 2a and Fig. 2b, for the low turbulence case ($Tu = 1\%$), a good match was achieved by precisely imposing experimental values at $x/C_{ax} = -2.3$, as the turbulence level remains approximately constant throughout the upstream length of the blade LE. Also, the HTC on the airfoil surface can be predicted with good accuracy using RANS approaches, since all tested models return a good matching with experiments (Fig. 2c). Even if the peak value is slightly overestimated, the CFDs are able to predict a second peak value at about 10% of the abscissa on the SS, while the HTC decay is a little steeper than measurements. The main difference between the models stays on the transition onset position on the SS since the $\gamma - Re_\theta$ model does not predict the transition and the flow stays laminar all along the SS, while the one-equation intermittency model predicts the transition at about 60% of the abscissa. Unfortunately, the lack of experimental data after 60% of abscissa does not allow to state which transition model performs better.

For the high turbulence case, the optimal boundary conditions were reached after multiple inlet intensity-eddy scale combinations were tested. In the present work, only the most representative ones are reported.



(a) $Tu = 1\%$ - Turbulence decay
(b) $Tu = 1\%$ - Length scale
(c) $Tu = 1\%$ - HTC
(d) $Tu = 15\%$ - Turbulence decay
(e) $Tu = 15\%$ - Length scale
(f) $Tu = 15\%$ - HTC

Fig. 2: Evaluation of (a)-(d) turbulence intensity, (b)-(e) length scale upstream the vane leading edge and (c)-(f) heat transfer coefficient on the vane surface with different CFDs approaches.



(a) $Tu_0 = 18\%$, $l_0 = 3.5mm$ - RANS (b) $Tu_0 = 18\%$, $l_0 = 12mm$ - RANS (c) $Tu_0 = 18\%$ and $l_0 = 12mm$ - LES

Fig. 3: Numerical turbulence intensity decay downstream the domain entrance for different inlet boundary conditions

All these simulations were performed with a $\gamma - Re_\theta$ transition model. As highlighted by Fig. 2d, setting the measured values of turbulence intensity and length scale at the inlet resulted in an underestimation of the experimental value at $x/Cax = -1.5$ (red line), as well as the entire trend, given by Roach [14] correlation for the adopted grid. Therefore, it became necessary to increase the characteristic eddies length scale at the inlet up to $12mm$ to limit the dissipation of vortexes associated with smaller scales. As already pointed out by Sarkar et al. [5], another possibility is to tune the C_μ constant of the turbulence model in addition to change intensity and length scale. Despite this setup resulting in turbulence decay in accordance with expectations without altering the measured length scale at the inlet, it is believed to be unreliable because changing the tuning constant of the model affects the resolution of the turbulent kinetic energy transport equation throughout the computational domain, leading to significant model drift and does not provide any general guideline. In other words, while modifying C_μ allows to match the turbulent kinetic energy decay upstream of the LE, the same modification leads to an incorrect evaluation of turbulent kinetic energy in the remaining domain regions. Specifically, as indicated by the sharp increase in turbulent length scale value in Fig. 2e, the dissipation of turbulent kinetic energy in the blade passage and downstream of it could be substantially underestimated alongside an excessive increase in the length scale. In the end, the most reliable setup for the high turbulence case is thought to be the one with $Tu_0 = 18\%$ and $l_0 = 12mm$ since it provides a good matching of the turbulence decay without neither a steep increase in eddies length scale (albeit higher values than measured ones) nor a modification of the $k - \omega$ model calibration constants. Such setup was then employed to assess the capability of γ transition model for HTC prediction and to initialize the inlet turbulence for the LES calculation.

Moving on to the HTC results, Fig. 2f collects the spanwise averaged profiles for all the simulation at high turbulence. As far as RANS calculations with the selected conditions ($Tu_0 = 18\%$ and $l_0 = 12mm$) are concerned, a significant overestimation of the measured values (red and purple dashed lines). In particular, values are strongly overestimated on both LE and PS. On the suction side, the one-equation γ transition model is able to catch the transition position slightly better than $\gamma - Re_\theta$, even if both methods return a much quicker transition than experimental results, thus hinting to a slightly different mechanism, as already pointed out by Ferreira et al. [3]. However, concerning the LES simulation with $Tu_0 = 18\%$ and $l_0 = 12mm$ at the domain inlet, it is able to replicate a good turbulence decay and to provide HTC values consistent with the measurements. As highlighted by the green line in Fig. 2f, the peak value at the LE and the trend on the PS are accurately captured. Also on the SS, the comparison with experimental data is quite good with a second peak lower than RANS predictions. On the other hand, no transition onset is predicted, showing a significant mismatch with experiments in this area. This is thought to be related to the computational grid, which would require a further refinement to better meet the requirements for a wall-resolved LES. It is worth noticing that imposing the measured turbulence intensity and length scale at the domain inlet of the RANS simulation (red solid line), without considering its decay along the inlet duct results in a better comparison with experiments; however, this is merely a consequence of underestimating the turbulence intensity at the vane inlet and along the passage, as shown by Figure 3.

7. Conclusions

The current work investigates the effect of free-stream turbulence intensity on the heat transfer around the academic LS-89 test case. Attention was put on imposing the correct boundary conditions at the domain inlet to accurately predict the experimental turbulence decay upstream of the vane leading edge at high values of free-stream turbulence. For a given inlet turbulence intensity, the turbulence length scale needs to be increased beyond experimental values to limit the dissipation associated with smaller-scale vortexes. Regarding the heat transfer predictions, given a similar turbulence decay between RANS and LES with the same inlet setup, the scale-resolved simulation successfully matches the experimental data, unlike the RANS approach. On the other hand, imposing the experimental boundary conditions leads to an underestimation of turbulence; the better agreement in terms of HTC between experiments and simulations is merely coincidental and could lead, in general, to erroneous conclusions.

References

[1] N. Gourdain, L. Gicquel, and E. Collado. "Comparison of RANS and LES for prediction of wall heat transfer in a highly loaded turbine guide vane". In: *Journal of propulsion and power* 28.2 (2012), pp. 423–433.

[2] J.C. Hoarau, P. Cinnella, and X. Gloerfelt. "Large Eddy Simulation of Turbomachinery Flows Using a High-order Implicit Residual Smoothing Scheme". In: *Computer and Fluids* 198 (2019), p. 104395.

[3] T.S. Cação Ferreira, T. Arts, and E. Croner. "On the Influence of High Turbulence on the Convective Heat Flux on the High-Pressure Turbine Vane LS89". In: *International Journal of Turbomachinery, Propulsion and Power* 4.4 (2019). ISSN: 2504-186X.

[4] I. Torrano, M. Tutar, M. Martinez-Agirre, A. Rouquier, and N. Mordantand M. Bourgoin. "Comparison of Experimental and RANS-Based Numerical Studies of the Decay of Grid-Generated Turbulence". In: *Journal of Fluids Engineering* 137.6 (June 2015), p. 061203. ISSN: 0098-2202.

[5] D. Sarkar E. and Savory. "Numerical Modeling of Freestream Turbulence Decay Using Different Commercial Computational Fluid Dynamics Codes". In: *Journal of Fluids Engineering* 143.4 (Feb. 2021), p. 041503. ISSN: 0098-2202.

[6] T. Arts and M. Lambert de Rouvroit. "Aero-Thermal Performance of a Two-Dimensional Highly Loaded Transonic Turbine Nozzle Guide Vane: A Test Case for Inviscid and Viscous Flow Computations". In: *Journal of Turbomachinery* 114.1 (Jan. 1992), pp. 147–154. ISSN: 0889-504X.

[7] T. Bacci, A. Picchi, T. Lenzi, B. Facchini, and L. Innocenti. "Effect of Surface Roughness and Inlet Turbulence Intensity on a Turbine Nozzle Guide Vane External Heat Transfer: Experimental Investigation on a Literature Test Case". In: *Journal of Turbomachinery* 143.4 (Mar. 2021), p. 041006. ISSN: 0889-504X.

[8] F.R. Menter. "Two-equation eddy-viscosity turbulence models for engineering applications". In: *AIAA journal* 32.8 (Aug. 1994), pp. 1598–1605. ISSN: 0001-1452.

[9] Robin Langtry and Florian Menter. "Transition modeling for general CFD applications in aeronautics". In: *43rd AIAA Aerospace Sciences Meeting and Exhibit*. Reno, Nevada: American Institute of Aeronautics and Astronautics, Jan. 2005.

[10] F.R. Menter, P. Smirnov, T. Liu, and R. Avancha. "A one-equation local correlation-based transition model". en. In: *Appl. Sci. Res.* 95.4 (Dec. 2015), pp. 583–619.

[11] F. Nicoud and F. Ducros. "Subgrid-Scale Stress Modelling Based on the Square of the Velocity Gradient Tensor". In: *Flow, Turbulence and Combustion* 62.3 (Sept. 1999), pp. 183–200.

[12] M.L. Shur, P.R. Spalart, M.K. Strelets, and A.K. Travin. "Synthetic Turbulence Generators for RANS-LES Interfaces in Zonal Simulations of Aerodynamic and Aeroacoustic Problems". In: *Flow, Turbulence and Combustion* 93.1 (July 2014), pp. 63–92.

[13] I.B. Celik, Z.N. Cehreli, and I. Yavuz. "Index of Resolution Quality for Large Eddy Simulations". In: *Journal of Fluids Engineering* 127.5 (Sept. 2005), pp. 949–958. ISSN: 0098-2202.

[14] P.E. Roach. "The generation of nearly isotropic turbulence by means of grids". In: *International Journal of Heat and Fluid Flow* 8.2 (1987), pp. 82–92. ISSN: 0142-727X.