

Analysis of Turbulence and Loss Mechanisms in Turbomachinery Cascades Using High-Fidelity Simulations

Analyse der Turbulenz- und Verlustmechanismen in Turbomaschinenkaskaden anhand von hochauflösenden Simulationen

Von der Fakultät für Maschinenwesen der
Rheinisch-Westfälischen Technischen Hochschule Aachen
zur Erlangung des akademischen Grades eines
Doktors der Ingenieurwissenschaften genehmigte Dissertation

vorgelegt von

Nima Fard Afshar

Berichter: Univ.-Prof. Dr.-Ing. Peter Jeschke
Univ.-Prof. Dr.-Ing. Dragan Kožulović

Tag der mündlichen Prüfung: 12.12.2025

Diese Dissertation ist auf den Internetseiten der Universitätsbibliothek online verfügbar.

DOI: <https://doi.org/10.18154/RWTH-2026-01454>

Kurzfassung

Die vorliegende Dissertation untersucht Turbulenz- und Verlustmechanismen in Kaskadenströmungen mithilfe hochauflösender numerischer Simulationen. Detaillierte hochgenaue Simulationen werden an einer hochbelasteten Niederdruckturbinenkaskade (NDT) ($Re = 90.000$) und einer Hochdruckverdichterkaskade (HDV) ($Re = 492.000$) durchgeführt, um die maßgeblichen physikalischen Mechanismen der Turbulenz- und Verlustentstehung zu identifizieren und zu quantifizieren. Ein besonderer Fokus liegt auf der Wechselwirkung zwischen Freistromturbulenz (FST), Transition und der Verlustentstehung in Kaskadenströmungen unter Off-Design-Bedingungen.

Beide Testfälle wurden als dreidimensionale Konfigurationen unter Berücksichtigung der Seitenwände mit hochauflösenden Methoden simuliert. Die Analyse konzentriert sich dabei auf das Strömungsverhalten in der mittleren Spannweite, wobei experimentelle Messungen (Profildruckverteilung und Nachlaufverluste) zur Validierung herangezogen werden. Fortgeschrittene statistische Methoden werden eingesetzt, um die dominanten Turbulenzstrukturen zu charakterisieren. Die Spektral- und Modal-Analysen zeigen, dass in der NDT niederfrequente Strukturen mit Transitions- und Ablösungsprozessen verbunden sind, während hochfrequente Komponenten turbulenten Strukturen in der wiederangelegten Strömung entsprechen. Die HDV weist dominante hochfrequente Strukturen innerhalb der Grenzschicht auf, die durch die starke Wechselwirkung zwischen FST und der druckgradienteninduzierten Ablösung entstehen. Eine Quadrantenanalyse liefert zusätzliche Einblicke in turbulente Ausmischprozesse innerhalb der Grenzschicht. In den Grenzschichten der beiden Kaskaden tragen die Ausstoß- und Sweep-Ereignisse am meisten zu den Ausmischprozessen bei. In der NDT-Kaskade reduziert eine erhöhte FST von $TI = 8\%$ die Ablöseblase um 20% und verringert die Gesamtverluste um 13% . Die Hauptverlustquellen befinden sich in der abgelösten Scherschicht und der wiederangelegten Strömung. Reynolds-averaged Navier-Stokes (RANS) Simulationen erfassen die Transition nicht genau und prognostizieren 32% geringere integrale Nachlaufverluste im Vergleich zu Large Eddy Simulationen (LES). In der HDV-Kaskade bewirkt eine erhöhte FST ($TI = 6\%$) eine frühere Transition um 10% der axialen Länge. Die Totaldruckverluste im Nachlauf ändern sich jedoch nicht signifikant, da die Mischungsprozesse in den verschiedenen Fällen ähnlich verlaufen. Die größten Verluste treten in der Grenzschicht auf der Profilsaugseite auf. Das RANS-Modell erfasst die laminare Ablösung nicht korrekt und überschätzt die integralen Nachlaufverluste um 55% im Vergleich zu LES.

Die Ergebnisse liefern wertvolle Informationen zur Validierung und Verbesserung von RANS-Turbulenzmodellen. Durch eine detaillierte Analyse der Turbulenzstrukturen trägt diese Arbeit zur Weiterentwicklung der Modellierung von Turbomaschinenströmungen bei, was letztlich zu Effizienzsteigerungen und einer Optimierung von Turbinen und Verdichtern führen kann. Die Ergebnisse zeigen, dass hochauflösende Simulationen als virtuelle Experimente genutzt werden können, um Defizite bestehender Modelle zu adressieren und zugleich experimentelle Untersuchungen zu ergänzen.

Abstract

In this dissertation, turbulence processes and flow loss mechanisms in turbomachinery cascade flows are analyzed through the application of high-fidelity numerical methods. Detailed simulations are conducted of a highly loaded low-pressure turbine cascade ($Re = 90,000$) and a high-pressure compressor cascade ($Re = 492,000$) to identify and quantify the key physical mechanisms contributing to turbulence and flow losses. A particular focus is placed on the interaction between free-stream turbulence (FST), transition processes, and flow loss generation in cascade flows under off-design conditions.

These two test cases were simulated in a three-dimensional configuration, including side-walls, using high-fidelity methods. However, the investigation primarily focuses on the mid-span flow behavior. The numerical computations are validated against experimental measurements of blade pressure distributions and wake flow losses. Advanced statistical techniques are applied to characterize the dominant turbulence structures. Spectral and modal analyses reveal that in the low-pressure turbine (LPT) cascade, low-frequency structures are associated with transitional and separated flow, whereas high-frequency components correspond to the development of turbulent structures during reattachment. The high-pressure compressor (HPC) cascade exhibits dominant high-frequency structures within the boundary layer, driven by the strong interaction between FST and pressure-gradient-induced separation. Quadrant analysis provides additional insights into turbulent exchange processes within the boundary layer. In the boundary layer flow of both the LPT and HPC cascades, ejection and sweep events are the biggest contributors to turbulent exchange processes. In the LPT cascade, increased FST-intensity $TI = 8\%$ enhances boundary layer mixing, leading to a 20% shorter separation length and a 13% reduction in total flow losses. The main flow loss sources are found in the separated shear layer and reattachment area, where turbulent kinetic energy accumulates. Reynolds-averaged Navier-Stokes (RANS) simulations fail to capture the transition dynamics accurately and underestimate the integral wake flow losses by 32% compared to large eddy simulations (LES). In the HPC cascade, an increased FST-intensity ($TI = 6\%$) induces an earlier transition of about 10% axial length, however, the total wake flow losses remain largely unchanged due to similar wake mixing characteristics across different FST-intensity cases. The most significant flow losses occur in the boundary layer near the suction side of the blade. The RANS model fails to capture the laminar separation accurately, overpredicting the integral wake flow losses by 55% relative to LES.

The findings provide valuable insights for validating and improving low-order RANS turbulence models. By offering a detailed analysis of turbulent structure formation, this work contributes to enhancing turbomachinery flow modeling, ultimately leading to efficiency improvements and performance optimization of turbines and compressors. The study demonstrates that high-fidelity simulations can be used as virtual experiments to address deficiencies in existing models and complement targeted experimental investigations.

Contents

1	Introduction	1
2	Theory	5
2.1	Linear cascades	5
2.2	Definition of Turbulence	6
2.2.1	State of Turbulence: Turbulence Anisotropy	9
2.2.2	Statistical methods	11
2.3	Navier-Stokes equations	13
2.3.1	Reynolds-averaged Navier-Stokes equations	14
2.3.2	Reynolds-Stress and turbulent kinetic energy equation	16
2.4	Discretization methods of the equation system	19
2.5	Simulation Strategies	20
2.5.1	DNS	21
2.5.2	LES	22
2.5.2.1	Explicit LES	23
2.5.2.2	Implicit LES (iLES)	23
2.5.3	RANS	25
2.5.3.1	$k - \omega$ turbulence model	26
2.5.3.2	$k - \omega$ - SST turbulence model	27
2.5.3.3	$k - \omega$ - Viscous-Blending turbulence model	28
2.5.4	Transition Modeling	29
2.6	Transition Over Turbomachinery Blades	31
2.6.1	Transition Types	33
2.6.2	Effect of free-stream turbulence on transition	37
2.7	Flow Losses	40
2.7.1	Turbine Flow - Transition and Flow Losses	43
2.7.2	Compressor Flow - Transition and Flow Losses	44
2.8	Post Processing Methods	47
2.8.1	Modal Analysis	47
2.8.1.1	POD	48
2.8.1.2	SPOD	48
2.8.2	Quadrant Analysis	49
3	Literature Review, Motivation and Outline of the Thesis	51
3.1	Literature review	51
3.1.1	Role of high-fidelity studies in turbomachinery	51
3.1.2	Turbine Cascade Flow Physics	52
3.1.3	Compressor Cascade Flow Physics	55
3.1.4	Comparison of high-fidelity methods with low order models	56
3.1.5	Flow loss analysis using high-fidelity methods	58

3.2	Motivation	59
3.3	Outline of the thesis	60
4	Numerical Methods	63
4.1	Spatial discretization	63
4.1.1	Finite Volume discretization	64
4.1.2	High order meshes	64
4.1.3	Spatial discretization requirements	65
4.2	Boundary condition	66
4.2.1	3D inflow conditions including side-walls	67
4.3	Temporal discretization	69
4.4	Error analysis	69
4.5	Test case description: Low pressure turbine cascade	73
4.5.1	Validation	75
4.6	Test case description: High pressure compressor cascade	78
4.6.1	Validation	78
5	Numerical Results of a Highly Loaded Low Pressure Turbine Cascade	83
5.1	Time-averaged results at mid-span	83
5.1.1	Surface analysis at mid-span	83
5.1.2	TKE analysis at mid-span	85
5.1.3	Total pressure loss at wake plane	87
5.2	Further Analysis of LES results	88
5.2.1	Anisotropy	88
5.2.2	Entropy analysis	91
5.2.3	Spectral analysis	94
5.2.4	Modal analysis	98
5.2.5	Quadrant Analysis	101
5.3	Influence of the free-stream turbulence on the flow at mid-span	104
5.4	Challenges of RANS in predicting turbine flows	112
5.5	A note on computational cost	114
6	Numerical Results of a High-Pressure Compressor Cascade	117
6.1	Time-averaged results at mid-span	117
6.1.1	Surface analysis at mid-span	117
6.1.2	TKE analysis at mid-span	119
6.1.3	Total pressure loss at wake plane	120
6.2	Further Analysis of LES results	122
6.2.1	Anisotropy	122
6.2.2	Entropy analysis	123
6.2.3	Spectral analysis	127
6.2.4	Quadrant analysis	132
6.3	Influence of the free-stream turbulence on the flow at mid-span	134
6.4	Challenges of RANS in predicting compressor flows	141
6.5	A note on computational cost	143
7	Summary and Outlook	145

Nomenclature	149
References	153
Appendix	175
A Modal Analysis	175
A.1 Proper Orthogonal Decomposition (POD)	175
A.2 Spectral Proper Orthogonal Decomposition (SPOD)	177
B $k - \varepsilon$ turbulence model	179
C $\gamma - \text{Re}_\theta$ transition model	179
D Error analysis	181

1 Introduction

Turbomachinery flows are characterized by complex interactions between stationary and rotating blades, subjected to elevated free-stream turbulence. These interactions generate a wide range of unsteady turbulent phenomena, including separation, mixing, laminar-to-turbulent transition, and flow reattachment, all of which significantly impact aerodynamic performance. Accurately predicting and controlling these phenomena is crucial for enhancing the efficiency of gas turbines and aero-engines, which play a key role in the decarbonization of propulsion and power generation. Standard turbulence modeling approaches, such as steady/unsteady Reynolds-averaged Navier-Stokes (RANS/URANS) simulations, remain the industry standard due to their computational efficiency. However, they struggle to accurately capture unsteady effects because periodic and transient disturbances are predominantly modeled rather than resolved. As modern turbomachinery designs push towards higher loadings and more compact configurations, the role of unsteadiness in determining performance becomes increasingly critical. A comprehensive understanding of turbulence and transition, along with advancements in turbulence modeling, is therefore essential for the development of more efficient and reliable engine designs.

Despite significant progress, further efficiency gains are increasingly difficult to achieve due to the aerodynamic maturity of existing designs. High-fidelity computational fluid dynamics (CFD) simulations, such as large eddy simulations (LES) and direct numerical simulations (DNS), offer the potential to overcome current modeling limitations by resolving a larger portion of the turbulence spectrum. These approaches, collectively referred to as high-fidelity simulations, allow for a more accurate representation of transition, turbulence, and flow loss mechanisms, providing insights that can enhance low-order turbulence models. However, the high computational cost of such simulations has limited their widespread application in industrial design processes. Recent advancements in high-performance computing (HPC) and numerical techniques are gradually making high-fidelity simulations more accessible. The emergence of large-scale parallel computing architectures has enabled detailed simulations of industrially relevant turbomachinery configurations, bridging the gap between academic research and industrial application. While the cost of full-scale multi-stage LES remains prohibitive, high-fidelity studies on simplified cascades or single-stage configurations provide valuable insight into fundamental flow physics (Gourdain et al. (2014)). By serving as virtual experiments, these simulations help refine turbulence models and improve predictive capabilities for industry-relevant flows. Furthermore, high-fidelity simulations offer a detailed assessment of turbulence anisotropy, entropy generation, and spectral characteristics, all of which are critical for understanding flow loss mechanisms in turbines and compressors. In particular, the spectral and modal analyses enable the identification of dominant turbulence structures across various frequency ranges, providing insight into transitional and separated flow behavior in low-pressure turbines and boundary layer interactions in high-pressure compressors. In

the long term, such virtual experiments may complement or even replace costly experimental campaigns by providing physical insights that are otherwise difficult or impossible to obtain through direct measurements.

This work contributes to the understanding of turbulence in off-design, transitional turbomachinery flows, focusing on a highly loaded low-pressure turbine and a modern high-pressure compressor cascade using high-fidelity methods. Through systematic analysis of turbulence development, transition mechanisms, and associated aerodynamic flow losses, along with their interplay across the passage, boundary layer, and wake region, this work provides fundamental insights that support the development of improved turbulence models and inform blade design optimization. Since most flow losses in turbine and compressor flows originate in boundary layers, understanding the influence of laminar-to-turbulent transition is vital for improving efficiency. Flow phenomena such as laminar-to-turbulent transition and separation remain particularly challenging for RANS turbulence and transition models. This study demonstrates the difficulties and limitations of RANS models in accurately capturing transitional flows at off-design operating points, specifically at low Reynolds numbers in turbines and under strong adverse pressure gradients in compressors. For instance, while RANS models fail to accurately predict the transition dynamics, leading to significant underestimation or overestimation of integral wake flow losses in low pressure turbine and high pressure compressor, LES provides a more accurate representation of flow loss mechanisms by resolving turbulence structures at multiple scales and therefore provides a more reliable flow loss representation. In this work, an alternative flow loss analysis method (other than the conventional total pressure loss analysis) is used to evaluate aerodynamic flow losses. This method is tailored to turbomachinery applications. Addressing the modeling deficiencies of RANS is essential for enhancing predictive accuracy and guiding the development of next-generation turbulence models. Furthermore, accurate prediction and mitigation of flow separation are essential for improving the overall aerodynamic performance of compressor and turbine blades. High-fidelity simulations reduce flow prediction uncertainties, potentially allowing for smaller safety margins in component operability.

In this work, the discrepancies between high-fidelity and low-fidelity simulations are investigated, and their respective predictive capabilities in modeling transition and aerodynamic flow losses in turbomachinery cascades are evaluated. A particular emphasis is placed on the differences in turbulent kinetic energy (TKE) modeling and their impact on flow loss prediction. Additionally, the influence of free-stream turbulence (FST) on boundary layer development and wake flow losses is explored through a series of high-fidelity simulations under varying turbulence intensities. To address these challenges, this work investigates the following key research questions. First, how do high-fidelity and low-fidelity methods compare in predicting transition and flow losses in transitional turbomachinery cascade flows at off-design points? Second, what are the dominant sources of aerodynamic flow losses in high-pressure compressors and low-pressure turbines? Third, how does free-stream turbulence affect transition mechanisms and flow losses in turbomachinery cascades?

While the influence of free-stream turbulence (FST) on profile aerodynamics and flow losses has been studied previously, this thesis extends the investigation by analyzing two industrially relevant cascades under realistic boundary conditions, including free-stream

turbulence and side-wall effects, at two distinct off-design operating points. In doing so, this work differentiates itself from previous research.

Numerical simulations are conducted using the TRACE (Turbomachinery Research Aerodynamics Computational Environment) solver (Becker et al. (2010), Edmund Kügeler (2005)), including high-fidelity and industry-relevant RANS methods. TRACE is jointly developed at the department of numerical methods at the Institute of Propulsion Technology of the German Aerospace Center (DLR) and MTU Aero Engines AG. Advanced statistical techniques, such as spectral and modal analysis, are applied to characterize turbulence behavior. The spectral analysis distinguishes between low- and high-frequency turbulent structures in transitional and fully turbulent flow regimes, while quadrant analysis provides a quantitative assessment of turbulent interactions in the boundary layer, highlighting the dominant ejection and sweep events that contribute to aerodynamic loss mechanisms. Although three-dimensional simulations (including side-walls) are performed, this study primarily focuses on the mid-span flow. However, three-dimensional modeling was necessary due to the diverging side-walls in the low-pressure turbine cascade and the low aspect ratio of the high-pressure compressor cascade. The high-fidelity results are validated against measurements of blade pressure distribution and wake flow losses, two key design metrics, to evaluate the validity and suitability of high-fidelity methods for design applications. Moreover, the high-fidelity results are compared against best-practice RANS simulations to highlight the deficiencies of RANS models in predicting blade pressure distribution and wake flow losses. Special attention is given to identify the sources of discrepancies between RANS turbulence models and high-fidelity simulations, especially regarding transition and flow loss prediction.

In the following chapters, the fundamental principles of turbulence, laminar-to-turbulent transition, and flow loss mechanisms in turbomachinery flows are presented. These concepts are then applied to explain the characteristics of boundary layer flow, wake flow, and the influence of free-stream turbulence observed in this work. The chapter concludes with a presentation of the numerical methods relevant to this study. A review of the current literature on high-fidelity simulations in turbomachinery flows follows, placing this work within the broader research context and outlining the research hypotheses. The numerical methods employed in this study are then described in detail, with a particular focus on high-fidelity simulations and the corresponding error analysis. The remaining part of this work presents and discusses the numerical results for each test case, validating the results against measurements and describing the turbulence phenomena and flow losses occurring in each case. Additionally, the effect of free-stream turbulence on laminar-to-turbulent transition, boundary layer, and wake flow losses is examined.

Both cascade geometries studied in this thesis were provided by MTU Aero Engines AG. All results are presented in a normalized form. This research was carried out as part of the national aviation research program V-3, funded by the German Federal Ministry for Economic Affairs and Energy (funding reference 20T1719D).

2 Theory

In the current chapter, the theoretical concepts relevant to this work are presented. First, the key flow features of linear cascades, as investigated in the current study, are highlighted. Next, turbulence is described as a phenomenon, and its fundamental properties are detailed. The statistical methods employed in the analysis are then introduced. Following this, the Navier-Stokes equations, along with further relevant transport equations, are reviewed. The various transition types and mechanisms that occur in turbomachinery flows are also discussed, including a detailed description of the separation bubble phenomenon and the associated separation-induced transition. Finally, the flow loss mechanisms in turbine and compressor cascades, along with its corresponding transport equation, is presented.

2.1 Linear cascades

The turbomachinery flow is in general very complex. It is therefore common to reduce the complexity for more detailed analysis. A significant simplification of axial turbomachinery is achieved by dividing the blade rings into coaxial cylinder sections. Under the assumption that the idealized flow of a coaxial cylinder surface differs only slightly from the flow in a real turbomachine in the mid-section, it is possible to reduce a three-dimensional problem to a two-dimensional problem. The simplified geometry can be well investigated both numerically and experimentally. The aforementioned configuration is referred to as a linear cascade. This simple configuration makes it easier to investigate certain mechanisms and their influence on the aerodynamic properties of the turbomachinery blades in isolation. Depending on the geometry under investigation, a pure two-dimensional representation is justified.

Blade rows of low-pressure turbines (LPTs) generally exhibit large spanwise dimensions relative to the chord length, resulting in moderate to high aspect ratios (3-7) that increase toward the rear stages. At high aspect ratios (above 2), the influence of secondary flows and side-wall boundary layers is largely confined to the end-wall regions, allowing the mid-span flow to be treated as approximately two-dimensional. Under these conditions, experimental and numerical investigations of LPT cascades are typically representative of real LPT blade rows, particularly with respect to mid-span aerodynamics and loss mechanisms.

On the other hand, compressor blade rows are characterized by lower aspect ratios and predominantly decelerating flow, which enhances the growth of side-wall boundary layers. Consequently, secondary-flow effects and end-wall losses extend further toward the mid-span region. This increased three-dimensionality significantly affects the axial velocity density ratio (AVDR), which becomes strongly dependent on the aspect ratio and the

development of the side-wall boundary layers. As a result, the mid-span flow in compressor cascades is more strongly influenced by end-wall effects, limiting the direct transferability of cascade measurements to real compressor blade rows unless three-dimensional flow interactions are explicitly accounted for.

Linear compressor cascade

The flow through the linear compressor cascade is characterized by a deceleration of fluid and therefore a rise in static pressure and static temperature. The flow in the linear compressor cascade passage resembles the flow in a diffuser. The flow on the suction surface of the linear compressor cascade is accelerated until the smallest cross-section of the blade profile and decelerated afterward because of the expansion of the cross-section.

Linear turbine cascade

On the other hand, the flow through the linear turbine cascades is dominated by the acceleration of the fluid, as a consequence, a reduction of static pressure and static temperature. This behavior is due to the expanding cross-section of the flow passage. The flow on the suction side of the linear turbine cascades encounters a significant acceleration, reaching a maximum velocity (e.g. Mach number), suction side peak before gradually decelerating to the same velocity of the pressure side at trailing edge. The aerodynamic design of the turbine blade defines the acceleration of the flow. The most important aerodynamic challenges in linear turbine cascades, especially the low-pressure turbine cascades, include boundary layer transition (e.g. separation), and the corresponding flow loss generation. The velocity distribution along the suction and pressure surface of compressor and turbine cascades, is influenced by factors such as incidence angle and free-stream turbulence. One conventional way is to plot the isentropic Mach number (M_{is} cf. (4.6)) over the blade surfaces instead of plotting the pressure or velocity distribution. In compressor flows, the suction side peak is largely affected by the incidence angle as shown in Fig. 2.1. From Fig. 2.1 it is visible how the positive incidence increases the blade loading and the negative incidence decreases the blade loading. In both cases, the acceleration region, on the suction surface, is largely influenced by the incidence variation.

Throughout this study, the isentropic Mach number distribution plot is utilized to characterize the flow over both the suction and pressure surfaces of turbine and compressor cascades. In the current work, one low-pressure turbine cascade and one high-pressure compressor cascade are investigated. Both geometries are investigated as three-dimensional geometry models including the side-walls. The reason for this choice is the diverging side-walls, in the low-pressure turbine cascade, and the low aspect ratio, in the high-pressure compressor cascade.

2.2 Definition of Turbulence

Defining turbulent flow concisely is a challenging task. However, several key characteristics can be identified. Turbulent flows are inherently irregular and stochastic in nature, which makes a purely deterministic description ineffective and, in many cases, infeasible. As a result, statistical methods are typically employed to describe and analyze such flows. One defining property of turbulence is its diffusivity, which leads to rapid mixing and enhanced rates of momentum, heat, and mass transfer. Turbulence is also inherently

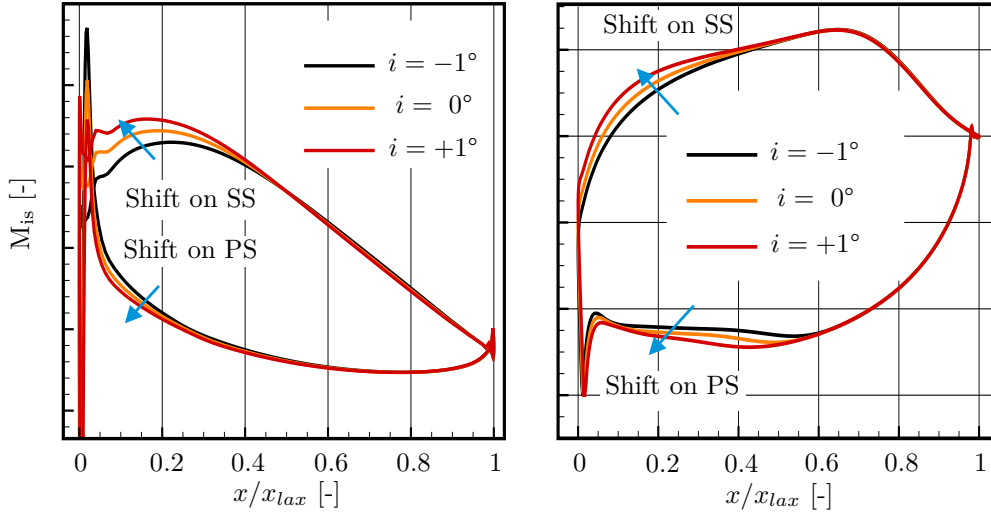


Figure 2.1: Influence of incidence angle on the isentropic Mach number (cf. (4.6)) distribution over compressor (left) and turbine (right) cascade blades, based on CFD results.

three-dimensional and rotational, characterized by intense vorticity fluctuations. The dynamics of vorticity are central to understanding turbulent flow behavior. A flow that appears random but does not exhibit dispersion or the spreading of velocity fluctuations into the surrounding fluid cannot be classified as turbulent (Entlesberger et al. (2005)). A key non-dimensional parameter governing the onset of turbulence, which expresses the ratio of inertial to viscous forces in a flow is the Reynolds number

$$\text{Re} = \frac{\rho c d}{\eta}, \quad (2.1)$$

where ρ is the density of the flow, c the magnitude of velocity, d a length characteristic of the flow, and η is the dynamic viscosity. Turbulence typically arises in flows with high Reynolds numbers, where the inertial forces dominate. The transition from laminar to turbulent flow is driven by instabilities that emerge when Re exceeds a critical threshold. These instabilities are the result of interactions between viscous damping and the nonlinear inertial terms in the Navier–Stokes equations.

Understanding the laminar-to-turbulent transition is fundamental to turbulence research. In the following sections (cf. Sec. 2.6), various transition mechanisms will be examined in more detail. Once turbulence is fully developed, it becomes essential to understand how turbulent energy is produced, transported, and dissipated. Turbulent flows are inherently dissipative, meaning viscous shear stresses convert turbulent kinetic energy into internal energy. Therefore, to sustain turbulence, a continuous input of energy is required, otherwise, the flow will decay. Turbulence occurs across a wide spectrum of spatial and temporal scales and is often described as a nonlinear superposition of vortical structures, ranging from large to very small eddies. These vortices interact through a process known as the energy cascade, first described by Richardson (1922). In this process, large eddies break down into progressively smaller structures due to shearing and entrainment of

surrounding fluid. As energy transfers from large to small scales, it eventually dissipates due to viscous effects at the smallest scales. Although energy typically cascades in the forward direction, from large to small eddies, it can also flow in the reverse direction in a process known as backscatter, where small-scale motions energize larger scales (Pope (2000), Müller-Schindewolf (2022)).

The turbulent energy spectrum (cf. Fig. 2.2) provides a useful framework to characterize this scale interaction in statistically steady, homogeneous turbulence. In the absence of external influences such as mean shear, the spectral energy distribution can be decomposed into three distinct regions (Pope (2000))

1. Energy-containing range hosts the largest eddies, where most of the turbulent kinetic energy resides. These structures are influenced by boundary conditions and flow geometry and are typically resolved in unsteady Reynolds-averaged Navier–Stokes (URANS) simulations.
2. In inertial subrange, energy is transferred across scales with negligible dissipation. This region follows Kolmogorov’s $\kappa^{-5/3}$ power law and is a key target for large eddy simulations (LES), where only the smallest subgrid scales are modeled.
3. In dissipation range, at the smallest scales, energy is dissipated into internal energy due to viscosity. These motions can only be resolved in direct numerical simulations (DNS), which remain computationally feasible primarily for low Reynolds number or academic flows.

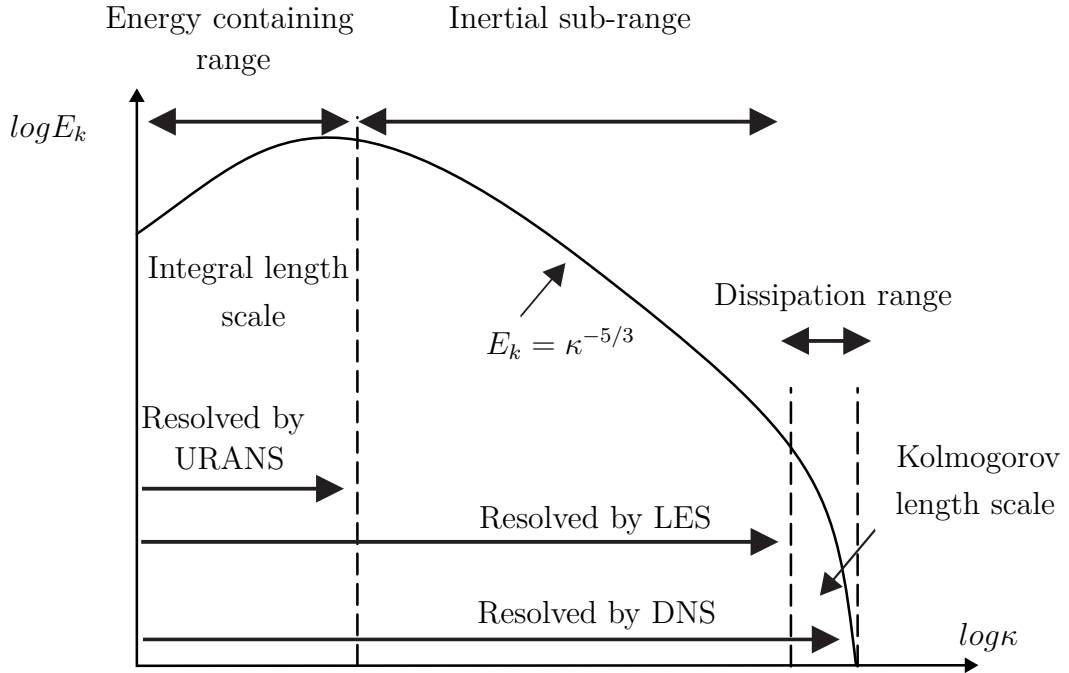


Figure 2.2: Three-dimensional model energy spectrum (turbulent energy cascade). E_κ represents the energy spectrum and κ represents the wavenumber.

In isotropic turbulence, the inertial subrange appears as a straight line with a slope of $\kappa^{-5/3}$ in a log-log plot of the energy spectrum E_κ , where κ denotes the wavenumber. The

smallest length scale in the turbulence spectrum, beyond which viscous effects dominate, is the Kolmogorov length scale

$$\eta = \left(\frac{\nu^3}{\varepsilon} \right)^{\frac{1}{4}}, \quad (2.2)$$

where ε being the specific dissipation and ν the kinematic viscosity. Similarly, the Kolmogorov time scale is defined as

$$\tau_\eta = \left(\frac{\nu}{\varepsilon} \right)^{\frac{1}{2}}. \quad (2.3)$$

Kolmogorov scales provide fundamental reference values for assessing grid resolution in high-fidelity simulations. In DNS, all scales down to η must be fully resolved, requiring extremely fine computational grids and accurate boundary conditions to solve the Navier–Stokes equations without modeling assumptions.

This spectral framework is critical for selecting appropriate turbulence modeling strategies based on computational resources and the level of physical fidelity required. Figure 2.3 schematically illustrates the unsteady phenomena in turbomachinery and highlights the limitations of various methods, particularly in capturing stochastic, turbulence-driven behavior. Figure 2.3 illustrates the phenomena called the spectral gap, which posits that deterministic and stochastic unsteadiness exist on separate scales without significant interaction. The assumption of spectral gap is a critical limitation of URANS methods.

Turbulence anisotropy is first examined, as it represents one of the defining and most important characteristics of turbulence.

2.2.1 State of Turbulence: Turbulence Anisotropy

Any arbitrary turbulence state can be reconstructed by a linear combination of the three limiting turbulence states, namely one-component (1C), isotropic two-component (2C), and isotropic three-component (3C). Isotropic 3C turbulence, as the name suggests, forces the anisotropy to be zero, also called "spherical-shape" turbulence. Isotropic 2C turbulence represents a turbulence state in which only two fluctuating components of equal intensity exist ("pancake-shape" turbulence). In contrast, one-component turbulence (1C) is characterized by one fluctuating component which dominates over the other two (e.g. grid turbulence through axisymmetric expansion) and is called "cigar-shape" turbulence. The turbulence states and the corresponding shapes are illustrated as a sketch in Fig. 2.4.

Flows in engineering applications are mostly strongly non-equilibrium and anisotropic due to boundaries, rotations and high adverse pressure gradients. Turbulence anisotropy can be quantified by incompressible form of the Reynolds stress anisotropy tensor

$$a_{ij} = \frac{\overline{c'_i c'_j}}{2TKE} - \frac{\delta_{ij}}{3}, \quad TKE = k = \frac{\overline{c'_i c'_i}}{2}, \quad (2.4)$$

where TKE is the turbulent kinetic energy, δ_{ij} the Kronecker delta and c'_i velocity fluctuation. The Reynolds stress anisotropy tensor describes the deviation of the Reynolds stress tensor from the isotropic state and can be interpreted as its non-dimensional deviatoric

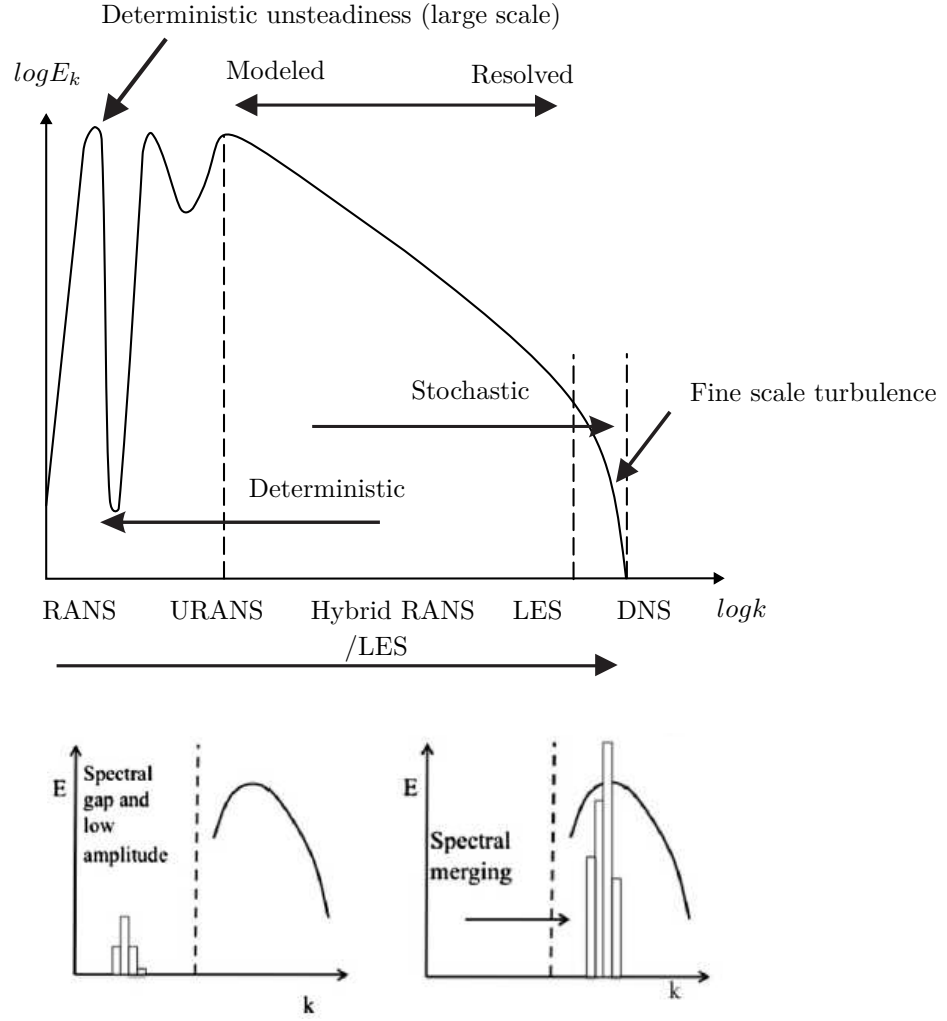


Figure 2.3: Three-dimensional model energy spectrum (turbulent energy cascade) with the corresponding turbulence models according to Sandberg et al. (2019). Spectral gap required for theoretically valid URANS from Scillitoe (2017)

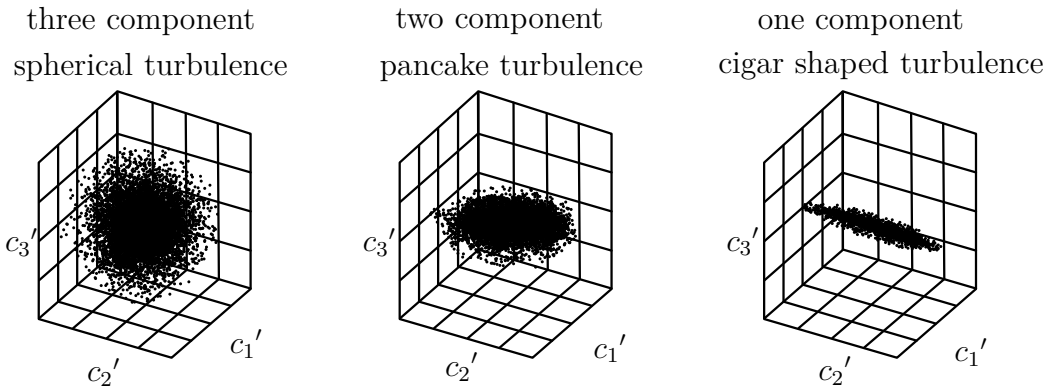


Figure 2.4: Anisotropy states

part. The definition of anisotropy allows one to measure the misalignment between the eigenvectors of Reynolds stress tensor and mean strain rate tensor. However, this quantity depends on the chosen coordinate system. Therefore, invariants of the anisotropy tensor, which are independent of the coordinate system, are an appropriate measure to model

turbulence in a frame-invariant way. The invariants of the anisotropy tensor can be determined from the anisotropy tensor (a_{ij}) or its eigenvalues λ . The eigenvalues λ_1 , λ_2 and λ_3 satisfy the ordering $\lambda_1 \geq \lambda_2 \geq \lambda_3$ and the constraint $\lambda_1 + \lambda_2 + \lambda_3 = 0$. By design, the first invariant, the trace of a_{ij} , is zero. The invariants can be defined as

$$I = a_{ii} = 0 \quad (2.5)$$

$$A_2 = II = a_{ij}a_{ji} = \lambda_1^2 + \lambda_1\lambda_2 + \lambda_2^2 \quad (2.6)$$

$$A_3 = III = a_{ij}a_{jk}a_{ki} = -\lambda_1\lambda_2(\lambda_1 + \lambda_2). \quad (2.7)$$

The weights of the limiting states of the turbulence are given by

$$C_{1c} = \frac{1}{2}(\lambda_1 - \lambda_2) \quad (2.8)$$

$$C_{2c} = \lambda_2 - \lambda_3 \quad (2.9)$$

$$C_{3c} = 3/2(\lambda_1) + 1, \quad (2.10)$$

where $C_{1c} + C_{2c} + C_{3c} = 1$. Based on the limiting states of turbulence, a so-called turbulence triangle can be defined (cf. Fig. 2.5). The disadvantage of the turbulence triangle is that the progression between the limit states is not linear due to the unequal weighting. Due to the non-linearity, flow states often appear to be closer to a boundary state than they really are. To avoid this problem, barycentric coordinates (x_b and y_b in (2.11)) are used. This utilizes the fact that each turbulence state is a linear, convex combination of the three corner states and can be defined

$$x_b = C_{1c}x_{1c} + C_{2c}x_{2c} + C_{3c}x_{3c}, \quad y_b = C_{1c}y_{1c} + C_{2c}y_{2c} + C_{3c}y_{3c}. \quad (2.11)$$

Figure 2.5 (right) shows a sketch of the barycentric map including the color map. The three corners correspond to pure red, green and blue, whereas the colors in between are blended. In this work the barycentric map will be used as the preferred color map to visualize the distribution of anisotropy within the flow in contour plots.

2.2.2 Statistical methods

The complex nature of turbulence, characterized by its randomness, nonlinearity, and multiscale behavior, makes it one of the most challenging unsolved problems in physics. The development of general solutions to the governing nonlinear partial differential equations remains elusive, and current theoretical efforts are largely based on heuristic and empirical approaches. Due to the inherently random nature of turbulent flow, statistical tools are indispensable for meaningful analysis. In this section, some of these statistical methods are introduced.

Probability theory provides the mathematical means to describe randomly distributed variables. The order of statistical moments m of a random variable x_n is defined by (2.12)

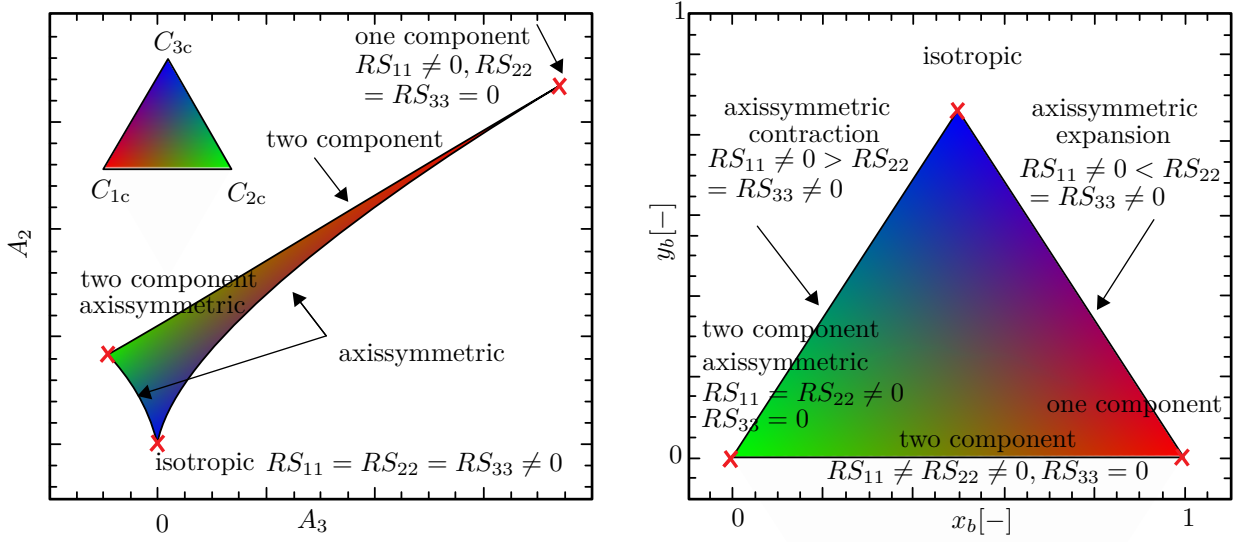


Figure 2.5: Lumley invariant map colored with two component parameter (left). Barycentric representation of turbulence states colored with RGB (right). The boundaries and corners confine any state of turbulence. The distribution of the Reynolds stresses at the boundaries are given exemplary. RS corresponds to Reynolds stress. A_2 and A_3 are the invariants of the Reynolds stress anisotropy tensor. x_b and y_b are the barycentric coordinates.

$$\overline{x^m} = \lim_{N \rightarrow \infty} \frac{1}{N} \sum_{n=1}^N x_n^m \quad (2.12)$$

$$\bar{x} = \lim_{N \rightarrow \infty} \frac{1}{N} \sum_{n=1}^N x_n \quad (2.13)$$

$$\sqrt{\overline{x^2}} = \lim_{N \rightarrow \infty} \sqrt{\frac{1}{N} \sum_{n=1}^N x_n^2}. \quad (2.14)$$

Here N is the number of realizations. The first order statistical moment of a random variable is therefore the mean value and can be calculated as in (2.13). The root of the second-order moment is the root mean square (RMS) and is calculated as in (2.14). When analyzing random variables, it is often practical to use central moments that consider the deviation from the mean. The central moment of order m is defined by

$$\overline{x'^m} = \lim_{N \rightarrow \infty} \frac{1}{N} \sum_{n=1}^N (x_n - \bar{x})^m \quad (2.15)$$

$$\overline{x'} = 0 \quad (2.16)$$

$$\overline{x'^2} = s. \quad (2.17)$$

In case of first order moment, the value of central moment is by definition zero as shown in (2.16). The second-order moment is the so-called variance s (2.17), which is closely

linked to the standard deviation as in

$$\sigma = \sqrt{s}, \quad (2.18)$$

which is a measure of the variance around the expected value. When considering the velocity fluctuations, the variance is proportional to the turbulent kinetic energy (TKE e.g. k) bounded in each velocity component. The sum of velocity components, therefore, corresponds to the turbulent kinetic energy of the flow as calculated in (2.4). Throughout this work, turbulent kinetic energy (TKE), is extensively used to characterize turbulence. The standard deviation of the fluctuating velocity components corresponds to the intensity of the turbulence, e.g. turbulence intensity (TI) described by

$$TI = \frac{\sqrt{\frac{1}{3}c_1'^2}}{\bar{c}} = \frac{\sqrt{\frac{2}{3}k}}{\bar{c}}. \quad (2.19)$$

In the case of isotropic turbulence, in which the statistical properties of the turbulence are identical in all spatial directions, it can be assumed that $\overline{c_1'^2} = \overline{c_2'^2} = \overline{c_3'^2}$. This assumption results in $TI = \sqrt{\overline{c_1'^2}}/\bar{c}$. The third and fourth statistical moments will not be used in this work and therefore will not be discussed further. Due to the statistically stationary nature of the flow in linear cascades, main aspects of engineering interest in the analysis of high-fidelity simulation results are first- and second-order statistical moments. Quantities such as the averaged isentropic Mach number around the blade profile, pressure losses in a wake cut or Reynolds stresses and their budget terms are typically of interest and used to validate the results against experimental data or to compare them with RANS simulations.

2.3 Navier-Stokes equations

The flow in turbomachines is described by the three-dimensional compressible Navier-Stokes equations, derived from the fundamental principles of mass, momentum, and energy conservation within the framework of continuum mechanics. The Navier-Stokes equations for the compressible flows can be expressed as

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho c_j)}{\partial x_j} = 0 \quad (2.20)$$

$$\frac{\partial (\rho c_i)}{\partial t} + \frac{\partial (\rho c_i c_j)}{\partial x_j} = -\frac{\partial p_s}{\partial x_i} + \frac{\partial \tau_{v,ij}}{\partial x_j} \quad (2.21)$$

$$\frac{\partial \left[\rho \left(e + \frac{c_i u_i}{2} \right) \right]}{\partial t} + \frac{\partial \left[\rho c_j \left(h + \frac{c_i c_i}{2} \right) \right]}{\partial x_j} = \frac{\partial (c_i \tau_{v,ij})}{\partial x_j} - \frac{\partial q_j}{\partial x_j}. \quad (2.22)$$

When formulating the equations, Einstein's summation convention (Einstein (1916)) is used, as for all other transport equations in this work. In the above equations, the static pressure is denoted by p_s , the viscous stress tensor by $\tau_{v,ij}$, the specific internal energy by

e , the specific enthalpy by h and the heat flux vector by q_j . Furthermore, the variables x_i and c_i are the individual components of the position vector $\vec{x}$ and the velocity vector $\vec{c}$. On the left-hand side of the above equations is the material derivative of the variable, which describes the change in the quantity with time t ($\frac{\partial}{\partial t}$) and through convection ($c \cdot \frac{\partial}{\partial x}$). In the case of the momentum equation (2.21), the material derivative of the momentum ρc_i is in equilibrium with the sum of the acting surface forces due to pressure and friction. Other acting forces such as the volume force density due to gravity are not taken into account in this work. In the energy conservation equation (2.22), external source terms, such as the power due to radiation or gravity, are already neglected. The Navier-Stokes equations represent a system of non-linear partial differential equations of the second order, for which there is no exact analytical solution except for a few special cases.

The pressure p is related to the other thermodynamic variables through the equation of state for an ideal gas

$$p = (\gamma - 1)\rho e, \quad (2.23)$$

where, γ , is the ratio between the specific heats of the fluids and is chosen $\gamma = 1.4$ for all the simulations in this work. The total enthalpy is given by $H = E + p/\rho$. The stress tensor of a Newtonian fluid can be expressed as

$$\tau_{v,ij} = \mu \left(\frac{\partial c_i}{\partial x_j} + \frac{\partial c_j}{\partial x_i} - \frac{2}{3} \frac{\partial c_k}{\partial x_k} \delta_{ij} \right). \quad (2.24)$$

In this case, the shear-rate tensor can be defined as

$$S_{ij} = \frac{1}{2} \left(\frac{\partial c_i}{\partial x_j} + \frac{\partial c_j}{\partial x_i} \right). \quad (2.25)$$

The dynamic viscosity μ is calculated with the help of the Sutherland's law. The heat flux vector q is defined as

$$q_i = -\lambda \frac{\partial T}{\partial x_i}, \quad i = 1, 2, 3. \quad (2.26)$$

Additionally, the dynamic viscosity μ is related to conductivity λ via

$$\text{Pr} = \frac{c_p \mu}{\lambda} = 0.72, \quad (2.27)$$

assuming a constant Prandtl number.

2.3.1 Reynolds-averaged Navier-Stokes equations

The Reynolds-averaged Navier-Stokes equations (RANS) result from the Navier-Stokes equations by applying the Reynolds decomposition, a statistical treatment of the fluctuations of the flow briefly introduced in the following. According to Reynolds (1895) the instantaneous flow quantities (ϕ) can be decomposed in a mean and a fluctuating part

$$\phi = \bar{\phi} + \phi'. \quad (2.28)$$

In case of compressible Navier-Stokes equations, a density-weighted averaging, called Favre average

$$\tilde{\phi} = \frac{1}{\bar{\rho}} \overline{\rho \phi}, \quad (2.29)$$

appears to be more appropriate. The averaging is performed for all quantities except pressure and density. The decomposition is written as

$$\phi = \tilde{\phi} + \phi''. \quad (2.30)$$

Applying the aforementioned decomposition to the Navier-Stokes equations, results in the following system of equations for the mean flow results

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \bar{\rho} \tilde{c}_i}{\partial x_i} = 0 \quad (2.31)$$

$$\frac{\partial \bar{\rho} \tilde{c}_i}{\partial t} + \frac{\partial \bar{\rho} \tilde{c}_j \tilde{c}_i}{\partial x_j} = -\frac{\partial \bar{p}_s}{\partial x_i} + \frac{\partial (\bar{\tau}_{v,ij} - \overline{\rho c_j'' c_i''})}{\partial x_j} \quad (2.32)$$

$$\begin{aligned} & \frac{\partial [\bar{\rho} (\bar{e} + \frac{1}{2} \overline{c_i c_i} + k)]}{\partial t} + \frac{\partial [\bar{\rho} \tilde{c}_j (\bar{h} + \frac{1}{2} \overline{c_i c_i} + k)]}{\partial x_j} = \\ & \frac{\partial}{\partial x_j} \left[-\bar{q}_j - \overline{\rho c_j'' h''} + \overline{\tau_{v,ij} c_i''} - \frac{1}{2} \overline{\rho c_j'' c_i'' c_i''} + \tilde{c}_i (\bar{\tau}_{v,ij} - \overline{\rho c_i'' c_j''}) \right]. \end{aligned} \quad (2.33)$$

RANS equations introduce new unknowns such as the Reynolds stress tensor $\overline{\rho c_i'' c_j''}$, describing the mean transport of momentum due to turbulence. This quantity acts similar as the viscous shear stress and therefore referred to as Reynolds stress. In order to solve the RANS equations, the set of equations need to be closed by introducing new correlations. This task is called the turbulence closure problem, which has been researched on for over a century and resulted in a very large number of turbulence models.

Most RANS turbulence models used in the engineering applications still use the Boussinesq assumption (Boussinesq (1877)). The Boussinesq assumption relates the Reynolds stress tensor linearly to the mean flow strain-rate tensor, with the help of a scalar quantity called the eddy (turbulent) viscosity (μ_t). The extensive use of the Boussinesq assumption in various turbulence models, created a class of models referred to as linear eddy viscosity models. Some of these widely-used models for turbomachinery applications will be explained in detail in Sec. 2.5.3. The Reynolds Stress tensor can be written as

$$\tilde{\rho} \widetilde{c_i'' c_j''} = \bar{\rho} \begin{pmatrix} \widetilde{c_1'' c_1''} & \widetilde{c_1'' c_2''} & \widetilde{c_1'' c_3''} \\ \widetilde{c_2'' c_1''} & \widetilde{c_2'' c_2''} & \widetilde{c_2'' c_3''} \\ \widetilde{c_3'' c_1''} & \widetilde{c_3'' c_2''} & \widetilde{c_3'' c_3''} \end{pmatrix} \quad (2.34)$$

The Reynolds Stress tensor is a symmetric tensor with $\overline{c_1' c_2'} = \overline{c_2' c_1'}$, $\overline{c_1' c_3'} = \overline{c_3' c_1'}$ and $\overline{c_3' c_2'} = \overline{c_2' c_3'}$. The entries on the main diagonal of the tensor correspond to the introduced variance of the velocity fluctuation and the trace of the main diagonal is equal to $2k$. The off-diagonal terms are called turbulent shear stress terms and are a measure of the shear forces occurring due to the turbulent momentum exchange.

2.3.2 Reynolds-Stress and turbulent kinetic energy equation

As already stated, the RANS equations lead to new unknown quantities. In case of the momentum equations, these are the six Reynolds stresses. For these, the transport equations are obtained by subtracting the RANS momentum equation (2.32) from the Navier-Stokes momentum equation (2.21). In this way, a transport equation can be written for each component of the symmetric Reynolds stress tensor as follows

$$\frac{\partial (\overline{\rho c_i'' c_j''})}{\partial t} + \frac{\partial}{\partial x_k} (\bar{c}_k \overline{\rho c_i'' c_j''}) = \bar{\rho} P_{ij} - \bar{\rho} \varepsilon_{ij} + \bar{\rho} \Pi_{ij} + \frac{\partial}{\partial x_k} (D_{ij,k}^\mu + D_{ij,k}^u + D_{ij,k}^p) + \bar{\rho} M_{ij} \quad (2.35)$$

$$\bar{\rho} P_{ij} = - \left(\overline{\rho c_j'' c_k''} \frac{\partial \bar{c}_i}{\partial x_k} + \overline{\rho c_i'' c_k''} \frac{\partial \bar{c}_j}{\partial x_k} \right) \quad (2.36)$$

$$\bar{\rho} \varepsilon_{ij} = \overline{\tau_{kj}'' \frac{\partial c_i''}{\partial x_k} + \tau_{ki}'' \frac{\partial c_j''}{\partial x_k}} \quad (2.37)$$

$$\bar{\rho} \Pi_{ij} = \overline{p_s \left(\frac{\partial c_i''}{\partial x_j} + \frac{\partial c_j''}{\partial x_i} \right)} \quad (2.38)$$

$$D_{ij,k}^\mu = \overline{\tau_{kj}'' c_i'' + \tau_{ki}'' c_j''} \quad (2.39)$$

$$D_{ij,k}^c = - \overline{\rho c_i'' c_j'' c_k''} \quad (2.40)$$

$$D_{ij,k}^p = - \overline{p_s c_i''} \delta_{jk} - \overline{p_s c_j''} \delta_{ik} \quad (2.41)$$

$$\bar{\rho} M_{ij} = - \left(\overline{c_i''} \frac{\partial \bar{p}_s}{\partial x_j} + \overline{c_j''} \frac{\partial \bar{p}_s}{\partial x_i} \right). \quad (2.42)$$

It is obvious that these equations contain new unknowns, such as the higher statistical moment $\overline{\rho c_i'' c_j'' c_k''}$ and therefore does not solve the turbulence closure problem. Setting up further transport equations for the new unknowns merely results in even more unknown quantities. Nevertheless the Reynolds Stress transport equations (2.35) provide us useful insights regarding the redistribution of the energy in turbulent flows. The right hand side of the equation consists of the source term $\bar{\rho} P_{ij}$, sink term $\bar{\rho} \varepsilon_{ij}$ and the redistribution terms of diffusion by turbulent transport $D_{ij,k}^c$, pressure fluctuation $D_{ij,k}^p$, molecular diffusion $D_{ij,k}^\mu$, turbulent mass flow by density fluctuation $\bar{\rho} M_{ij}$ (negligible for $M < 5$) and the pressure-shear term $\bar{\rho} \Pi_{ij}$.

Of particular importance for flows with anisotropic turbulence (as in turbomachinery flow) is the pressure-shear term ($\bar{\rho} \Pi_{ij}$). This term leads to a redistribution of energy between the turbulent normal stresses, which is known as the return-to-isotropy process and has a strong influence on the mixing of wakes (Batchelor et al. (1946)). The more pronounced the anisotropy, the greater the pressure shear terms, which equalize the anisotropy behavior.

By contracting (equating the indices i and j) of (2.35), the transport equation for the turbulent kinetic energy can be generated

$$\bar{\rho} \frac{\partial k}{\partial t} + \bar{\rho} \tilde{c}_j \frac{\partial k}{\partial x_j} = \bar{\rho} P - \bar{\rho} \varepsilon + \frac{\partial}{\partial x_j} (D_j^\mu + D_j^c + D_j^p) + \bar{\rho} M \quad (2.43)$$

$$\bar{\rho} P = -\overline{\rho c_i'' c_j''} \frac{\partial \tilde{c}_i}{\partial x_j} \quad (2.44)$$

$$\bar{\rho} \varepsilon = \overline{\tau_{ij}' \frac{\partial c_i''}{\partial x_j}} \quad (2.45)$$

$$D_j^\mu = \overline{\tau_{ij}'' c_i''} \quad (2.46)$$

$$D_j^c = -\frac{1}{\rho} \overline{\rho c_j'' c_i'' c_i''} \quad (2.47)$$

$$D_j^p = -\overline{p_s c_j''} \quad (2.48)$$

$$\bar{\rho} M = -\overline{c_i'' \frac{\partial \bar{p}_s}{\partial x_j}} + \overline{p_s \frac{\partial c_i''}{\partial x_i}}. \quad (2.49)$$

(2.43) describes the transport of the trace of the Reynolds stress tensor, i.e. the turbulent kinetic energy (TKE). Except for the missing pressure-shear term, the turbulent kinetic energy equation (2.43) has similar source, sink and redistribution terms as the transport equation of the Reynolds stresses (2.35). The production term P and the dissipation term ε are again source and sink terms. The terms of turbulent transport due to velocity fluctuations D_j^c and pressure fluctuations D_j^p are often also named as turbulent convection. In this case, the turbulent fluctuation energy and the pressure fluctuation are transported by the turbulent motion. The turbulent mass flow ($\bar{\rho} M$) consists of the pressure work and the pressure dilation.

The counterpart to the transport equation of mean turbulent motion (2.43) is the energy equation of mean motion

$$\begin{aligned} \bar{\rho} \frac{\partial \left(\frac{1}{2} \bar{c}_j^2 \right)}{\partial t} + \overline{\rho c_i} \frac{\partial \left(\frac{1}{2} \bar{c}_j^2 \right)}{\partial x_i} &= \frac{\partial}{\partial x_j} \underbrace{\left[-\overline{\rho c_i c_i' c_j'} - \bar{c}_j \bar{p}_s + \eta \bar{c}_i \left(\frac{\partial \bar{c}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right]}_{\text{Redistribution}} \\ &+ \underbrace{\overline{\rho c_i' c_j'} \frac{\partial \bar{c}_i}{\partial x_j}}_{\text{Production} - \bar{\rho} P} - \underbrace{\eta \frac{\partial \bar{c}_i}{\partial x_j} \left(\frac{\partial \bar{c}_i}{\partial x_j} + \frac{\partial \bar{c}_j}{\partial x_i} \right)}_{\text{Dissipation } \bar{\varepsilon}}. \end{aligned} \quad (2.50)$$

The production term of the turbulent kinetic energy appears in (2.50) with a positive sign and therefore usually corresponds to a sink. The production of the turbulent kinetic energy is therefore produced at the expense of the mean kinetic energy. The production is largely dependent on the shear rate S_{ij} and the turbulent shear stresses. However, the turbulent normal stresses also contribute to the production of turbulent kinetic energy in decelerated flows. An acceleration of the flow, on the other hand, causes a negative production term P and thus a reduction of turbulent kinetic energy in favor of the mean kinetic energy. Subsequently, the transport equation of the averaged total pressure is considered

$$\frac{\partial \bar{p}_t}{\partial t} - \frac{\partial \bar{p}_s}{\partial t} + \bar{c}_j \frac{\partial \bar{p}_t}{\partial x_j} = \underbrace{\eta \frac{\partial^2 \left(\frac{1}{2} \bar{c}_i^2 \right)}{\partial x_j^2}}_{\text{Redistribution}} - \underbrace{\frac{\partial \left(\overline{\rho c_i c'_i c'_j} \right)}{\partial x_j}}_{\text{Production} - \bar{\rho} \mathcal{P}} + \underbrace{\overline{\rho c'_i c'_j} \frac{\partial \bar{c}_i}{\partial x_j}}_{\text{Dissipation } \bar{\varepsilon}} - \eta \left(\frac{\partial \bar{c}_i}{\partial x_j} \right)^2, \quad (2.51)$$

as this quantity plays a major role in quantifying flow losses in compressible flows and is used extensively in this work.

This equation follows from the energy equation of the mean motion (2.50) assuming an incompressible flow and by inserting $\bar{p}_t = \bar{p}_s + 0.5 \bar{c}_i^2$. The production term of the turbulence appears here as a sink and is therefore responsible for a decrease in total pressure. This key relationship between turbulence production and total pressure loss will be extensively utilized in this work. As demonstrated in the following chapters, regions of high turbulence production correspond to areas of total pressure loss. Therefore, in this work, special emphasis will be placed on turbulence production, as it plays a significant role in flow loss generation.

The same equation (2.51) can be written in a more general sense, containing mean and turbulent components. To obtain the general transport equation of total pressure, same as in the transport equation of turbulent kinetic energy (2.43), momentum and continuity equations can be rearranged to obtain the following equation

$$\frac{Dp_t}{Dt} \approx \frac{D(\rho(1/2)c_i c_i)}{Dt} + c_i \frac{\partial p}{\partial x_j} = c_i \frac{\partial \tau_{ij}}{\partial x_j}. \quad (2.52)$$

By applying a Reynolds decomposition and subsequent time averaging, the following transport equation for total pressure

$$\begin{aligned} \underbrace{\bar{c}_i \frac{\partial \bar{p}_t}{\partial x_i}}_{Totalpressure_m} &\approx \underbrace{\bar{c}_i \frac{\partial}{\partial x_j} \left(\bar{\rho} \frac{1}{2} \bar{c}_i \bar{c}_i \right)}_{A_m} + \underbrace{\bar{c}_i \frac{\partial}{\partial x_j} \left(\bar{\rho} \frac{1}{2} \overline{c'_i c'_i} \right)}_{A_f} + \underbrace{\bar{c}_i \frac{\partial \bar{p}}{\partial x_i}}_{pwork_m} = \\ &- \underbrace{\bar{\tau}_{ij} \frac{\partial \bar{c}_i}{\partial x_j}}_{V_m} - \underbrace{\overline{\tau'_{ij}} \frac{\partial \overline{c'_i}}{\partial x_j}}_{V_f} - \underbrace{\frac{\partial}{\partial x_j} \left[\bar{c}_i \left(\overline{\rho c'_i c'_j} \right) \right]}_{tr_{mf}} - \underbrace{\frac{\partial}{\partial x_j} \left[\overline{\rho c'_j} \left(\frac{1}{2} \overline{c'_i c'_i} \right) \right]}_{tr_{ff}} \\ &+ \underbrace{\frac{\partial \bar{\tau}_{ij} \bar{c}_i}{\partial x_j}}_{vdiff_m} + \underbrace{\frac{\partial \overline{\tau'_{ij} c'_i}}{\partial x_j}}_{vdiff_f} + \underbrace{\bar{c}_i \frac{\partial \bar{p}}{\partial x_i}}_{pwork_f} - \underbrace{\frac{\partial \overline{p' c'_i}}{\partial x_i}}_{pdif_f} + \underbrace{\overline{p' \frac{\partial c'_i}{\partial x_i}}}_{pdil_f}, \end{aligned} \quad (2.53)$$

is obtained. Where $Totalpressure_m$ is the mean flow of total pressure, A is the advection of kinetic energy, $pwork$ is the pressure work, tr is the turbulent transport due to unsteadiness, $vdiff$ is the viscous diffusion, V is the viscous dissipation, $pdif$ is the pressure diffusion, and $pdil$ is the pressure dilation. The subscripts m and f represent mean and fluctuating contributions to each term. The above derivations are according to Przytarski et al. (2024).

2.4 Discretization methods of the equation system

In this section, an overview of the numerical methods commonly used in turbomachinery flow simulations is provided, with particular emphasis on the numerical approaches employed in this work.

Finite Difference

The finite difference (FD) method is one of the most traditional, straightforward and established techniques for approximating solutions to partial differential equations (PDEs). FD directly approximates spatial derivatives using truncated Taylor series expansions. The FD method can achieve higher accuracy by incorporating additional terms from the Taylor expansion, thereby involving more degrees of freedom (DOF) in each solution update. However, FD methods are generally constrained to structured grids, making them less flexible for complex geometries. Although FD methods can be implemented efficiently, they are generally limited to simple grids and geometries. Furthermore, FD schemes do not inherently ensure conservation unless special methods are applied, which becomes particularly challenging on curvilinear grids (Blazek (2005)).

Finite Volume

In contrast to FD methods, the finite volume (FV) method discretizes the integral form of the Navier-Stokes equations over control volumes (cells), ensuring conservation of mass, momentum, and energy. The Gauss theorem is employed to convert the convective and diffusive terms from volume to surface integrals, which are evaluated using fluxes across cell interfaces. The coupling between cells is typically handled using Riemann solvers, such as Roe's approximate solver. A first-order scheme is obtained when only adjacent cell values are considered at the interfaces, while higher-order accuracy can be achieved using extended stencils similar to FD methods. Despite its widespread use in simulating industrial flows, the FV method shares some limitations with FD methods, especially when applied to complex geometries or distorted meshes, which can degrade accuracy (Hirsch (2007) and LeVeque (2012)).

Finite Element

Originally developed for structural mechanics, the finite element (FE) method has also been successfully adapted for fluid dynamics. Similar to the FV method, the domain is divided into non-overlapping elements. The governing PDEs are formulated using a weighted residual approach, where a test function is multiplied with the equation and integrated over the domain. Through integration by parts, the divergence of the flux terms is removed. The weak solution is obtained by ensuring the weighted residuals vanish for every test function within the function space. The solution is typically expressed as a linear combination of basis functions, which are required to be continuous across element boundaries. A widely used variant is the Galerkin method, which applies the same basis for both the test function and solution approximation. The Galerkin method is commonly used with linear polynomials leading to second-order accuracy. However, conventional Galerkin methods can suffer from numerical oscillations in advection-dominated problems, leading to the development of optimized schemes or the addition of artificial diffusion (Zienkiewicz et al. (2014)).

Discontinuous Galerkin

A particularly effective class of FE methods for fluid dynamics is the discontinuous Ga-

lerkin method (DG-method). Unlike conventional FE methods, DG allows discontinuities at element boundaries, enhancing its ability to handle complex flow structures. By coupling elements via Riemann solvers and solving volume and surface integrals using quadrature rules, DG methods maintain accuracy and robustness even in challenging conditions, such as shock waves and turbulence. Discontinuous Galerkin spectral element method (DGSEM) is a specialized and highly efficient variant of the DG method, particularly well-suited for large-scale simulations requiring high accuracy. DGSEM uses nodal tensor-product Lagrange basis functions on hexahedral meshes with collocated quadrature nodes, typically using Legendre-Gauss (LG) or Legendre-Gauss-Lobatto (LGL) points. This collocation enables the omission of many computationally expensive operations found in conventional DG methods, resulting in significant efficiency gains. The compact, element-local stencil of DGSEM ensures excellent scalability for massively parallel computations, making it highly attractive for large-scale simulations.

DGSEM is particularly beneficial for implicit large eddy simulations (cf. Sec. 2.5.2.2), where numerical dissipation naturally regulates unresolved turbulence without requiring explicit sub-grid scale models. However, one of the main challenges of DGSEM is maintaining stability, especially in high-Reynolds-number turbulent flows. The use of collocated solution and quadrature nodes can introduce aliasing errors and nonlinear instabilities, which are particularly problematic in high-order schemes that aim to minimize numerical dissipation and assure high accuracy. To mitigate these stability issues, two primary techniques have been developed. Spatial filtering and over-integration (or polynomial de-aliasing), which removes unphysical content but introduces additional dissipation, potentially reducing the scheme's effective resolution. Over-integration, on the other hand, improves numerical accuracy by refining the flux integration process, though at an increased computational cost. Despite these improvements, challenges persist, particularly for extremely high Reynolds number flows. A significant advancement in the DGSEM framework is the development of a summation-by-parts (SBP) property, combined with split-form flux formulations. These techniques effectively reduce the aliasing errors and enhance numerical stability without excessively increasing dissipation. This approach has demonstrated remarkable success in highly turbulent flow simulations, where conventional methods often struggle (Gregor J. Gassner et al. (2016) and Bergmann (2023)).

In summary, DGSEM is distinguished by its combination of efficiency, scalability, and high-order accuracy. Despite challenges related to stability in turbulent flows, recent developments like the SBP property have made it a robust and powerful tool for fluid dynamics simulations. These attributes make DGSEM a suitable DG method especially for higher Reynolds number cases. In this work, both FV and DGSEM methods are employed to perform the simulations, leveraging their respective strengths for different flow scenarios.

2.5 Simulation Strategies

Turbulent and transitional flows exhibit complex characteristics. Consequently, various simulation strategies have been developed to address these challenges, each tailored to specific applications and research interests. The following sections will explore these approaches in detail.

2.5.1 DNS

Direct numerical simulation (DNS) is a computational technique that resolves all turbulence scales in a fluid flow by directly solving the Navier-Stokes equations without any modeling approximations. DNS aims to capture both spatial and temporal details of the flow with high accuracy, making it a powerful tool for studying turbulence. However, the computational cost of DNS is extraordinarily high, particularly as the Reynolds number increases, due to the necessity of resolving the smallest turbulent scales. The computational demands of DNS scale with the Reynolds number, as the ratio of the largest to smallest length scales in the flow grows. For homogeneous turbulence, Pope (2000) determined that the spatial resolution requirement scales with the Reynolds number (Re) to the power of $9/4$

$$n^3 \sim Re^{9/4}, \quad (2.54)$$

which, when including temporal resolution, becomes even more demanding. A more recent estimate by Choi et al. (2001) suggests an even steeper scaling

$$n^3 \sim Re^{37/14}. \quad (2.55)$$

This exponential growth in computational cost makes DNS impractical for high Reynolds number flows in large-scale engineering applications, such as an entire aircraft. Spalart et al. (1997) estimated that a DNS of the flow over a full commercial airliner would not be feasible before at least 2080, requiring a computational grid with 10^{16} degrees of freedom (DOF). More recent assessments indicate that even this estimate might be overly optimistic (Spalart et al. (2016), Spalart (2000) and Bergmann (2023)). Despite these challenges, the primary advantage of DNS is its accuracy and the absence of empirical modeling. Given appropriate boundary conditions, DNS provides an exact numerical solution to the governing equations, offering an unprecedented level of detail that is invaluable for turbulence research. As a result, DNS remains an essential tool for fundamental fluid dynamics studies (Bergmann (2023)).

With advances in computational power, DNS has become more feasible for smaller, less complex systems, such as individual components of engineering systems. For example, DNS has been successfully applied to mid-span flow of low-pressure turbine cascades (Wu et al. (2001) and Sandberg et al. (2015)), though often with some relaxation of strict resolution criteria, meaning not all turbulence scales are fully captured. Instead, these simulations aim to resolve the majority of the turbulent structures, often without the need for explicit turbulence models for the unresolved scales. However, the relaxation of these criteria raises concerns about the classification of certain DNS simulations reported in the literature. In some cases, the computational grid does not fully resolve down to the Kolmogorov scale, the smallest turbulence scale (cf. (2.2)), while non-optimized low-order discretization methods can introduce numerical dispersion errors. Consequently, some simulations labeled as DNS may more accurately be classified as implicit large eddy simulations (ILES) or under-resolved DNS, where only the most energy-containing scales are resolved.

2.5.2 LES

Large eddy simulation (LES) is a computational approach designed for flow problems where Reynolds-averaged Navier-Stokes (RANS) models lack sufficient accuracy, yet the computational cost of direct numerical simulation (DNS) is prohibitively high. As a part of the high-fidelity simulation family, LES directly computes turbulent fluctuations in both time and space. However, unlike DNS, which resolves all turbulence scales, LES focuses on capturing only the large, energy-containing eddies (cf. Fig. 2.2) while modeling the effects of the smaller, unresolved scales. In this regard, LES serves as an intermediate method between DNS and RANS. Instead of employing time averaging like RANS, in LES, a spatial filtering to derive a filtered set of Navier-Stokes equations is applied. This approach necessitates a turbulence model to close the equations and account for the effects of unresolved turbulent motions. Specifically, the subgrid-scale (SGS) model is used to represent the influence of eddies smaller than the grid resolution, while larger eddies are explicitly resolved. The filtering operation, whether explicitly defined or implicitly applied through the computational grid, effectively suppresses features smaller than the filter width, while larger scales remain unaffected (Sagaut (2006)).

A key advantage of LES is its ability to resolve most of the turbulence while modeling only the smallest scales, making it less dependent on empirical turbulence models compared to RANS. By retaining time dependency, LES provides a more accurate representation of physical phenomena, especially in flows dominated by large-scale unsteady motions. The velocity field is typically decomposed into resolved and residual (subgrid) components, ensuring that most energy-containing, anisotropic flow structures are directly captured. However, fine near-wall structures, such as streaks, require a high spatial resolution, increasing computational costs (Sagaut (2006) and Tyacke et al. (2014)). LES-based methods also offer improved physical accuracy in predicting boundary layer instabilities, which are essential for turbulence generation and sustenance. While some believe that simple models suffice for representing small-scale isotropic eddies, near-wall turbulence remains highly anisotropic, necessitating more sophisticated modeling approaches. The choice of turbulence model must account for the Reynolds number and the filtering technique used, as these factors significantly impact the accuracy of subgrid-scale modeling. LES remains a more computationally feasible alternative to DNS, enabling simulations of higher Reynolds number flows that would otherwise be impractical.

The SGS model plays a crucial role in accurately modeling turbulent stresses while maintaining control over flow dynamics. In well-resolved simulations, where the largest eddies contribute significantly to turbulent kinetic energy production, the SGS model extracts the appropriate energy, ensuring minimal influence on the resolved flow. However, in cases of insufficient resolution, errors primarily arise at scales close to the grid spacing, while the largest eddies remain well captured. SGS models are typically derived from either empirical knowledge or fundamental properties of the filtered Navier-Stokes equations, with the primary objective of representing the effects of sub-filter scales on the resolved flow (Chattot et al. (2009), Pino Martín et al. (2000) and Vreman et al. (1995)).

In the TRACE solver (used in this work), FV-LES calculations are performed by solving the filtered compressible Navier-Stokes equations using a second-order accurate, density-based finite volume scheme with MUSCL reconstruction (Sengupta et al. (2001) and Van Leer (1979)). The LES calculation employs a fraction of 10^{-3} of Roe's numerical flux

(Roe (1981)), which is added to a central flux to prevent odd-even decoupling. For time integration, a third-order accurate explicit Runge-Kutta method is used, ensuring temporal accuracy in the simulations presented in this thesis.

2.5.2.1 Explicit LES

The term explicit LES refers to models that apply an explicit formulation to approximate the subgrid-scale (SGS) terms. Over the years, various approaches have been developed to model the subgrid stress tensor (Pino Martín et al. (2000)). However, most of these models are still based on the Boussinesq hypothesis (cf. Sec. 2.5.3). The anisotropic component of the subgrid stresses can be approximated using the resolved flow variables

$$\tau_{ij}^{\text{SGS}} \approx \tau_{ij} - \frac{1}{3}\delta_{ij}\tau_{kk} \approx -2\bar{\rho}\nu_{\text{sgs}} \left(\hat{S}_{ij} - \frac{1}{3}\delta_{ij}\hat{S}_{kk} \right). \quad (2.56)$$

The term, ν_{sgs} , represents the sub-grid eddy viscosity. This term is determined from the resolved flow. As mentioned earlier, there are many subgrid scale models, varying in the complexity. In the current thesis, the wall-adapting local eddy viscosity (WALE) model, developed by Nicoud et al. (1999), has been used in combination with the FV discretization scheme to compute the subgrid stresses. In this formulation, the subgrid stresses are calculated as

$$\nu_{\text{sgs}} = C_w^2 \Delta^2 \frac{(\hat{S}_{ij}^d \hat{S}_{ij}^d)^{3/2}}{(\hat{S}_{ij} \hat{S}_{ij})^{5/2} + (\hat{S}_{ij}^d \hat{S}_{ij}^d)^{5/4}}, \quad (2.57)$$

where Δ is a characteristic filter width and $\hat{S}_{ij}^d$ is given by

$$\hat{S}_{ij}^d = \frac{1}{2}(\tilde{g}_{ij}^2 + \tilde{g}_{ji}^2) - \frac{1}{3}\delta_{ij}\tilde{g}_{kk}^2, \quad (2.58)$$

where $\tilde{g}_{ij} = \frac{\partial \tilde{c}_i}{\partial x_j}$ and $\tilde{g}_{ij}^2 = \tilde{g}_{ik}\tilde{g}_{kj}$.

The implementation used in this work is the version of the WALE model incorporated into the TRACE solver. In this formulation, the model constant $C_w = 0.5587$ has been calibrated with the FV scheme of TRACE on isotropic decaying turbulence experiments by Comte et al. (1996). This LES approach and implementation have been validated for several applications (Morsbach et al. (2020) and Leyh et al. (2020)).

2.5.2.2 Implicit LES (iLES)

As already mentioned, high-fidelity simulations often employ the high-order discontinuous Galerkin method within an implicit large eddy simulation (ILES) framework, where numerical dissipation replaces explicit subgrid-scale models. A widely used variant, the discontinuous Galerkin spectral element method (DGSEM) (Blaisdell et al. (1996), Kopriva (2009), Kopriva et al. (2002)), enhances computational efficiency by leveraging collocated solution and integration points (Hindenlang et al. (2012), Beck et al. (2014), Bergmann

et al. (2016)). When combined with explicit time integration, DGSEM offers geometric flexibility, high-order accuracy, and excellent parallelization, making it well-suited for turbomachinery simulations. In DG, the computational domain Ω is discretized into non-overlapping elements Ω_j , where the solution is approximated using a polynomial expansion

$$\mathbf{q}(\mathbf{x}, t)|_{\Omega_j} \approx \mathbf{q}_h(\mathbf{x}, t)|_{\Omega_j} = \sum_{n=0}^{N^3} \mathbf{q}_n(t) \phi_n^N(\mathbf{x}), \quad \forall \Omega_j \in \Omega, \quad (2.59)$$

where $\mathbf{q}_n$ are local degrees of freedom and ϕ_n^N are basis functions. Applying a Galerkin projection to the governing equations

$$0 = \partial_t \mathbf{q} + \nabla \cdot \mathbf{F} \approx \partial_t \mathbf{q}_h + \nabla \cdot \mathbf{F}_h, \quad (2.60)$$

leads to the weak formulation

$$\int_{\Omega_j} \partial_t \mathbf{q}_h \phi_m^N d\mathbf{x} + \oint_{\partial\Omega_j} (\mathbf{F}_h \cdot \mathbf{n}_{\partial\Omega_j}) \phi_m^N dS - \int_{\Omega_j} \mathbf{F}_h \cdot \nabla \phi_m^N d\mathbf{x} = 0. \quad (2.61)$$

Since DG does not enforce global continuity, the numerical flux $\mathbf{H}(\mathbf{q}^+, \mathbf{q}^-, \mathbf{n})$ approximates fluxes at element interfaces, ensuring consistency and stability

$$\mathbf{H}(\mathbf{q}, \mathbf{q}) = \mathbf{F}(\mathbf{q}) \cdot \mathbf{n} \quad , \quad \mathbf{H}(\mathbf{q}|_{\Omega_j}, \mathbf{q}|_{\Omega_k}) = -\mathbf{H}(\mathbf{q}|_{\Omega_k}, \mathbf{q}|_{\Omega_j}). \quad (2.62)$$

For advective fluxes, Roe's flux-difference splitting method (Roe (1981)) is used

$$\mathbf{H}^a(\mathbf{q}_h^+, \mathbf{q}_h^-, \mathbf{n}) = \frac{1}{2} [\mathbf{F}_h^a(\mathbf{q}_h^+) + \mathbf{F}_h^a(\mathbf{q}_h^-)] \cdot \mathbf{n} + \epsilon |\tilde{A}| (\mathbf{q}_h^+ - \mathbf{q}_h^-), \quad (2.63)$$

where $A = [\partial \mathbf{F}_h^a / \partial \mathbf{q}_h]$ is the Jacobian at the Roe average state, and ϵ introduces entropy correction.

For viscous fluxes, the Bassi-Rebay 1 (BR1) method (Bassi et al. (1997a)) introduces an auxiliary variable $\mathbf{S} = \nabla \mathbf{q}$, reformulating the problem into a first-order system

$$\partial_t \mathbf{q} + \nabla \cdot \mathbf{F}(\mathbf{q}, \mathbf{S}) = 0 \mathbf{S} - \nabla \mathbf{q} = 0. \quad (2.64)$$

This leads to the weak formulation

$$\mathbf{H}_h^v(\mathbf{q}_h, \mathbf{S}_h) = \frac{1}{2} [\mathbf{F}_h^v(\mathbf{q}_h^-, \mathbf{S}_h^-) + \mathbf{F}_h^v(\mathbf{q}_h^+, \mathbf{S}_h^+)] \cdot \mathbf{n}. \quad (2.65)$$

To improve efficiency, Lagrange polynomial interpolation is used in a nodal basis

$$\phi_n^N = l_i^N(x_1) l_j^N(x_2) l_k^N(x_3), \quad (2.66)$$

where x_1, x_2, x_3 are reference coordinates. This results in the semi-discrete system

$$\mathbf{M} \frac{\partial \tilde{\mathbf{q}}}{\partial t} + \mathbf{R}(\mathbf{q}) = 0, \quad (2.67)$$

with a diagonal mass matrix

$$\mathbf{M} = \text{diag} [\omega_0, \dots, \omega_N], \quad (2.68)$$

where $\{\omega_i\}_{i=0}^N$ are Gauss quadrature weights.

2.5.3 RANS

The foundation of modern industrial flow simulations is based on the Reynolds-averaged Navier-Stokes (RANS) equations. These equations describe the mean flow, meaning that turbulent structures are not directly resolved. Due to their lower mesh resolution requirements and the possibility of performing stationary simulations, RANS models offer a computationally efficient approach. However, they face the closure problem, as the Reynolds stresses cannot be determined explicitly from the RANS averaging process (Pope (2000), Leschziner (2015)).

As outlined by Pope (2000), turbulence models are required to approximate the mean Reynolds stress field as accurately as possible. The accuracy of this approximation directly influences the modeling error. RANS models can generally be classified into two main categories. The first category consists of eddy viscosity models, which approximate the effective viscosity of the fluid as the sum of the molecular viscosity (μ) and turbulent viscosity (μ_t). A fundamental assumption of many eddy viscosity models is the Boussinesq hypothesis (Boussinesq (1877)), which expresses the turbulent stress tensor in terms of the strain rate tensor

$$\bar{\tau}_{ij} + \bar{\tau}_{t,ij} = \mu \left(\frac{\partial \bar{c}_i}{\partial x_j} + \frac{\partial \bar{c}_j}{\partial x_i} \right) - \rho \overline{c'_i c'_j} = (\mu + \mu_t) \left(\frac{\partial \bar{c}_i}{\partial x_j} + \frac{\partial \bar{c}_j}{\partial x_i} \right), \quad (2.69)$$

$$\bar{\tau}_{t,ij} = -\rho \overline{c'_i c'_j} = \mu_t \left(\frac{\partial \bar{c}_i}{\partial x_j} + \frac{\partial \bar{c}_j}{\partial x_i} - \frac{2}{3} \frac{\partial \bar{c}_k}{\partial x_k} \right) - \frac{2}{3} \delta_{ij} \rho k \quad (2.70)$$

Various turbulence models have been developed to approximate the turbulent viscosity (μ_t). Most of these models compute the turbulent viscosity using transport equations. Depending on the number of transport equations solved, eddy viscosity models are categorized into zero-equation, one-equation, and two-equation models. Zero-equation models do not solve any transport equations for turbulent kinetic energy, while two-equation models additionally solve for the turbulent dissipation rate ε .

In contrast to the eddy viscosity models, the group of Reynolds stress models has been established, whose intention is to determine the Reynolds stress tensor $\bar{\tau}_{t,ij}$, without relying on the assumption of a turbulent viscosity. This approach allows for a more accurate representation of turbulence anisotropy, making RSMs particularly suitable for complex flow scenarios where turbulence anisotropy plays a dominant role. Although RSMs offer

improved accuracy compared to eddy viscosity models, they come with increased computational cost and numerical complexity. They are also prone to instabilities and are highly dependent on mesh quality, making them less practical for industrial applications (Leschziner (2015)). In recent years, the RSM models have not been successful in replacing the RANS linear eddy viscosity models. Since the group of Reynolds stress models is not investigated in the current work, the details are not discussed any further.

2.5.3.1 $k - \omega$ turbulence model

As a two equation eddy viscosity turbulence mode, the $k - \omega$ model introduced by Wilcox (1988), has been developed alongside the $k - \varepsilon$ (cf. Appendix B) model over the past decades. These two models share structural similarities and can be transformed into one another using the relation

$$\omega = \frac{\varepsilon}{\beta^* k}, \quad (2.71)$$

where ω represents the specific dissipation rate and β^* is a calibration constant. The transport equation for the specific dissipation rate is

$$\frac{D\rho\omega}{Dt} = P_\omega - D_\omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right], \quad (2.72)$$

with the production and destruction terms

$$P_\omega = \alpha \frac{\omega}{k} \bar{\tau}_{t,ij} \frac{\partial \bar{c}_i}{\partial x_j}, \quad D_\omega = \beta \rho \omega^2, \quad (2.73)$$

where α und β are the calibration constants. The turbulent viscosity is calculated using

$$\mu_t = \frac{\rho k}{\omega}. \quad (2.74)$$

The transport equation for the turbulent kinetic energy (B.1) is similar to the one in $k - \varepsilon$ model (cf. Appendix B)

$$\frac{D\rho k}{Dt} = P_k - D_k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right], \quad (2.75)$$

with

$$P_k = \bar{\tau}_{t,ij} \frac{\partial \bar{c}_i}{\partial x_j}, \quad D_k = \beta^* \rho \omega k. \quad (2.76)$$

From a practical perspective, the following properties can be addressed to the $k - \omega$ model.

1. Improved accuracy for near-wall flows, making it particularly useful for boundary-layer modeling.

2. Extensive use in turbomachinery simulations, where it has established itself as a standard turbulence model.
3. Challenges in prescribing the specific dissipation rate ω at walls, since ω approaches infinity at the wall, requiring special numerical treatment.

2.5.3.2 $k - \omega$ - SST turbulence model

The $k - \omega$ -shear stress transport (SST) model, introduced by Menter (1994), is an advanced turbulence model that combines the strengths of the $k - \omega$ and $k - \varepsilon$ models. The $k - \omega$ model is known for its robustness in boundary layers, accurately predicting velocity profiles, particularly in flows with pressure gradients and compressibility effects. In contrast, the $k - \varepsilon$ model is more effective in free-stream flows and shear layers, such as wake regions. To merge these advantages, Menter (1994) transformed the ε equation (B.2) into ω using (2.71). This transformation introduces a cross-diffusion term in the ω equation, which is activated by the F_1 function

$$\frac{D\rho\omega}{Dt} = P_\omega - D_\omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + (1 - F_1) \underbrace{2\rho\sigma_\omega \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}}_{\varepsilon\text{-cross diffusion}} \quad (2.77)$$

There is also a continuous transition between the model constants of the $k - \omega$ and $k - \varepsilon$ model parts. The model function F_1 therefore has the task of distinguishing between areas close to the wall and the free flow in order to utilize the specific advantages of the models. This is achieved using the equations

$$F_1 = \tanh \left(\arg_1^4 \right) \quad (2.78)$$

$$\arg_1 = \min \left[\max \left(\frac{\sqrt{k}}{0.09\omega y}, \frac{500\nu}{y^2\omega} \right), \frac{4\rho\sigma_{\omega 2}k}{C_D y^2} \right] \quad (2.79)$$

$$C_D = \max \left(2\rho\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}, 10^{-20} \right). \quad (2.80)$$

The $\tanh$ function in (2.78) ensures a continuous transition of F_1 between zero and one. $\arg_1$ tends strongly towards zero with increasing wall distance y due to the functional relationship $1/y$ and $1/y^2$, whereby (2.77) changes to the ε modeling. The third term in (2.79) contains the C_D cross-diffusion term from (2.77) limited to the positive value range and, according to Menter (1994), serves as a safeguard that $\arg$ approaches zero at the boundary layer edge.

The model described up to this point is referred to as the baseline model by Menter (1994). The SST modification also includes a limitation of the turbulent eddy viscosity to improve accuracy in flows with strong adverse pressure gradients, ensuring consistency with Bradshaw's hypothesis (Bradshaw (1967))

$$\mu_t = \frac{\rho a_1 k}{\max(a_1 \Omega, SF_2)}, \quad (2.81)$$

where F_2 is another blending function

$$F_2 = \tanh \left(\arg_2^4 \right), \quad (2.82)$$

with

$$\arg_2 = \max \left(2 \frac{\sqrt{k}}{0.09\omega y}, \frac{500\nu}{y^2\omega} \right). \quad (2.83)$$

Here, a_1 is a model constant and ω is the specific dissipation rate.

(2.83) is formed similarly to the expression from (2.79). Consequently, $\arg_2$ will assume large values near the wall and (2.81) is limited to $\mu_t = \rho a_1 k / \Omega$. Here, the assumption of Bradshaw (1967) is adhered to, according to which the wall shear stress $\tau_W = \rho a_1 k$ must be proportional to turbulent kinetic energy (TKE), k . Since for two-equation models $\tau_W = \mu_t \Omega$ applies, the limited value of μ_t can be derived from the aforementioned expression.

2.5.3.3 $k - \omega$ - Viscous-Blending turbulence model

The $k - \omega$ Viscous blending (VB) is an extension of the $k - \omega$ model, introduced by Bode et al. (2014). The primary motivation behind this extension is to ensure proper adjustability of turbulent decay behavior, which is controlled by the destruction term D_k in the $k - \omega$ model (2.73). Various studies (Martinstetter (2010), Bode et al. (2014)) have shown that turbulence length scales, determined via autocorrelation in wind tunnels, are typically of the order of $l_\Lambda \approx 2.0$ cm. In high-turbulence cases ($TI \approx 8.5\%$), such as those found downstream of an active turbulence grid, length scales can reach $l_\Lambda \approx 10$ cm, comparable to the chord length of the investigated cascades (Aufderheide et al. (2014)). Bode et al. (2014) have shown with numerical studies that, as in the experiments, the specification of a corresponding length scale ($l_{k-\omega} \approx l_\Lambda$) is also necessary for modeling the turbulent decay behavior. This was later confirmed by several authors Pichler et al. (2015) and Schwarzbach et al. (2018) by means of LES investigations.

The turbulent length scale in the $k - \omega$ model (Wilcox (1988)) is calculated by

$$l_{k-\omega} = k^{1/2} / \omega. \quad (2.84)$$

According to (2.84), larger length scales imply smaller ω values, which slow down turbulence decay. Conversely, small values of ω according to (2.74) lead to a high turbulent viscosity μ_t . However, these high values of μ_t lead to the prediction of excessive turbulent stresses and an overproduction of turbulence in sheared flows, as described by Bode et al. (2014). As a result, flow losses are overestimated. This can also incorrectly force premature laminar-turbulent transition at the leading edge of the cascade. To separately model the turbulent decay behavior in free-stream and boundary layer regions and specify the optimum $l_{k-\omega}$ in each case, Bode et al. (2014) introduced a modified specific dissipation rate in the free-flow region exclusively depending on the inlet variables (Mach number, Turbulence intensity and specific dissipation at inlet)

$$\omega_{FS} = \left(246.93 (M_1 + TI_1)^{2.12} \omega_1 \right). \quad (2.85)$$

This equation follows an analytical decay function from Baines et al. (1951). However, Bode (2018) later replaced this with a local function $\omega_{\text{FS}} = f(k)$, which depends only on the decay of turbulent kinetic energy (k). A blending factor is introduced to distinguish between free-stream and boundary layer turbulence

$$b_v = \min \left(\max \left(\left(\frac{\omega}{\omega_{\text{FS}}} \right), 0.1 \right), 1.0 \right). \quad (2.86)$$

This blending factor modifies the turbulent viscosity formulation (similar to the $k - \omega$ -SST model) (Bode (2018))

$$\mu_t = \frac{\rho a_1 k}{\max(a_1 \omega, b_v S)}, \quad (2.87)$$

in areas with high values of $\omega/\omega_{\text{FS}}$, the model switches from ω to the shear rate S . Similar to (2.81) from the $k - \omega$ -SST model, the assumption of Bradshaw (1967) is also taken into account here, whereby the vorticity ω was replaced by the shear rate S . In most parts of the boundary layer, $\Omega = S$, but for example in the region of the stagnation point, a strong compression of the fluid occurs, which leads to $S \gg \Omega$. The limitation of $k - \omega - \text{VB}$, according to Bode (2018), has a stronger effect at the stagnation point. The effect achieved here thus also corresponds to the logic of the correction of the stagnation point anomaly according to Kato and Launder (Kato et al. (1993)). The $k - \omega$ -SST model and the $k - \omega$ -VB model are very similar in their basic features. Unlike the $k - \omega$ -SST model, which relies on cross-diffusion for transition blending, the VB model employs viscous blending based on shear rate S , ensuring better performance in sheared and stagnation regions. Additionally, the $k - \omega$ -VB model uses a non-fully-local formulation in Bode et al. (2014), as the reference to the nearest entry edge is always established in (2.85). This contradicts a common modeling philosophy of modern RANS models for application to complex, three-dimensional test cases (Menter et al. (2004), Langtry et al. (2009)), but makes the $k - \omega$ -VB model according to Bode et al. (2015) and Bode et al. (2014) more robust and is particularly suitable for internal flows, such as multi-stage turbomachinery. This results from the fact that individual block groups can be delimited by artificial boundary conditions for each blade row and the model refers to new global reference variables (Ma_1 , TI_1 and ω_1). This central difference in the mode of operation between the $k - \omega$ -SST model and the $k - \omega$ -VB model was eliminated by Bode (2018) and the model refers, among other things, to the local turbulent kinetic energy, which only decays weakly at correspondingly high turbulent length scales $l_{k-\omega}$ according to the model philosophy.

Figure 2.6 shows the eddy viscosity modeled by different turbulence model for a low pressure turbine cascade.

2.5.4 Transition Modeling

Traditional turbulence models struggle to predict boundary layer transition accurately and often overestimate the onset of turbulence (Langtry (2006)). To address this, transition models introduce an intermittency factor γ that modulates the production of turbulent kinetic energy, ensuring a laminar boundary layer when $\gamma = 0$. One widely adopted approach is the $\gamma - \text{Re}_\theta$ model, initially developed by Menter et al. (2004) and later refined by Langtry (2006) and Langtry et al. (2009). This model employs two transport equations

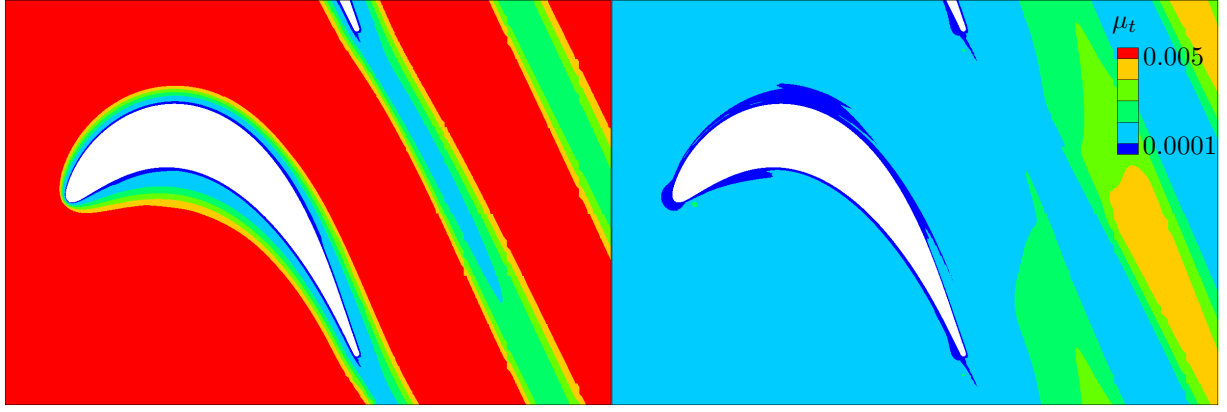


Figure 2.6: Eddy viscosity modeled by $k - \omega$ turbulence model (left) and $k - \omega$ -VB model (right)

for γ and critical Reynolds number ($Re_{\theta t}$), allowing it to track boundary layer history and predict transition when the local shear rate-based Reynolds number exceeds a critical threshold

$$Re_v = \frac{\rho y^2}{\mu} S. \quad (2.88)$$

The critical Reynolds number is determined using correlations from Abu-Ghannam et al. (1980), considering turbulence intensity (TI) and acceleration parameter (λ_θ)

$$Re_{\theta t} = f(TI, \lambda_\theta), \quad (2.89)$$

and the acceleration parameter (λ_θ) is

$$\lambda_\theta = \frac{\theta^2}{\nu} \frac{dc}{ds}. \quad (2.90)$$

In (C.4), θ is the momentum thickness and dc/ds is the acceleration along the streamwise direction and can be computed by taking the derivative of the velocity in the x, y and z directions and then summing the contribution of these derivatives along the streamwise flow direction (Langtry et al. (2009)). Unlike older methods requiring integration across the boundary layer, this model is fully local, making it well-suited for unstructured grids and parallel computing. Additionally, it can handle separation-induced transition by introducing γ_{sep} (cf. Appendix C), which accounts for shear layer behavior in separation bubbles. The transport equations for γ and $Re_{\theta t}$ (cf. Appendix C) include production and destruction terms that regulate transition onset and turbulent kinetic energy generation. This formulation enables accurate modeling of various transition mechanisms, including surface roughness, vortex-generating jets, and separation-induced effects, making it highly adaptable for aerodynamic applications. For a detailed formulation and additional derivations, refer to Appendix C.

2.6 Transition Over Turbomachinery Blades

The development of boundary layer theory was a milestone in fluid mechanics, enabling the understanding of laminar and turbulent flows. While computing these boundary layers has become relatively straightforward, accurately predicting laminar-to-turbulent transition remains challenging due to its inherently unstable, three-dimensional, and stochastic nature (Mayle (2018)). The transition process begins in a pre-transitional laminar region, progresses through an intermittent phase where laminar and turbulent patches coexist, and ends in a fully turbulent boundary layer. The onset of transition typically occurs at the location where isolated turbulent spots first form, primarily triggered by free-stream turbulence that amplifies upstream laminar fluctuations. This stage often precedes the minimum skin-friction coefficient (cf. 4.7) location. Conversely, the transition is considered complete when all remaining laminar structures vanish near the wall, usually downstream of the maximum skin-friction coefficient. A key challenge in transition modeling is not the computation of the pre-transitional flow, since existing models account for free-stream turbulence effects (Walters et al. (2004) and Walters et al. (2008)), but the accurate prediction of both transition onset and length. In turbomachinery flows, pre-transitional boundary layers extend from the leading edge to the point where turbulent spots first emerge, followed by an intermittent phase until fully turbulent conditions are established (Mayle (2018)). The extent of the transition region is primarily influenced by the pressure gradient and free-stream turbulence levels. A favorable pressure gradient or low disturbances prolong transition, while an adverse gradient or high disturbances accelerate it (Mayle (2018), Emmons (1951)). Understanding the interplay between external disturbances and boundary layer stability remains crucial for improving transition predictions in aerodynamic applications.

In the following section, key transition mechanisms in turbomachinery are explored, with a focus on pressure gradients and external disturbances, as these factors play a dominant role in the present work. The transition from laminar to turbulent flow is governed by multiple interacting factors, listed below.

Pressure Gradient

A negative static pressure gradient ($\frac{dp}{ds} < 0$), also called favorable pressure gradient, which corresponds to an accelerating flow, tends to stabilize the boundary layer, while the deceleration of the flow, which corresponds to a positive pressure gradient ($\frac{dp}{ds} > 0$), also called adverse pressure gradient, destabilizes it, promoting transition.

Figure 2.7 illustrates an indifference curve based on stability theory as presented by Schlichting et al. (2017), showing the Reynolds number range where boundary layer instabilities develop. The indifference Reynolds number (Re_{ind}) marks the threshold where disturbances begin to amplify. In other words, the area enclosed in the curves characterizes the area in which instabilities can form. For an accelerated boundary layer (i.e., where $\frac{dp}{ds} < 0$), this threshold is higher, meaning instabilities form at larger Reynolds numbers and further downstream. Boundary layers developing in an adverse pressure gradient have a point of inflection in the velocity profile, which is also shown in Fig. 2.7. Additionally, inviscid instabilities occur under positive pressure gradients and persist at high Reynolds numbers, where viscous effects diminish. This fact is characterized by the upper branch of the indifference curves for positive pressure gradients, which runs asymptotically towards a value other than zero at high Reynolds numbers (Müller-Schindewolf (2022)). The

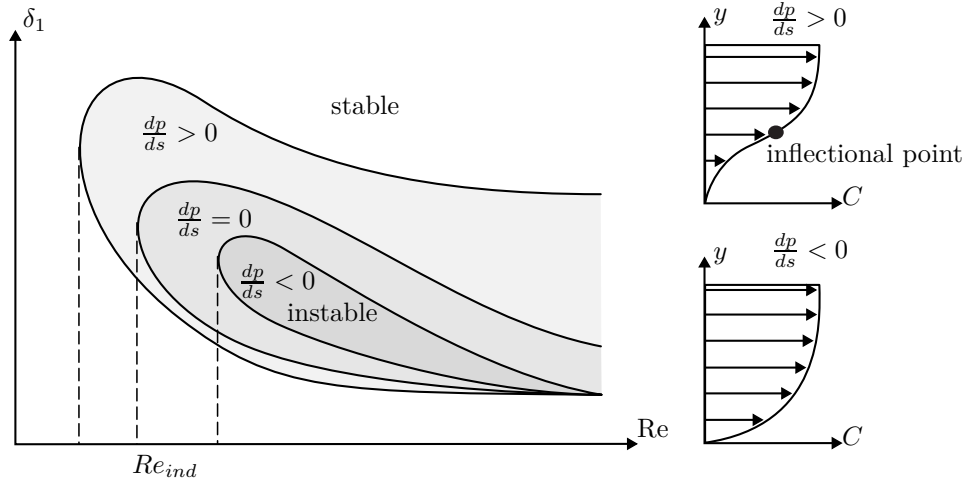


Figure 2.7: Schematic representation of an indifference curve for varying pressure gradients. The velocity profiles are shown for boundary layers under positive and negative pressure gradient. δ_1 refers to the boundary layer thickness.

sensitivity of transition to the pressure gradient is highly important for this work, as the flow over turbine and compressor cascades encounters varying pressure gradients across different operating regimes and over suction and pressure surfaces.

Reynolds number

The transition occurs only when the Reynolds number exceeds the critical indifference Reynolds number (Re_{ind}). Below this threshold, frictional forces dampen disturbances, maintaining flow stability.

Turbulence Intensity and Integral Length Scale

Free-stream turbulence can penetrate the boundary layer, and higher turbulence intensity generally favors transition by introducing disturbances. The integral length scale characterizes the size of these disturbances. A small integral length scale leads to rapid turbulence decay and delays transition, while only disturbances of a certain size are effectively amplified within the boundary layer. Figure 2.8, according to Saric et al. (2002), shows the course of the transition under the influence of external disturbances on the boundary layer as a function of the amplitude of the disturbance (increasing from left to right). At very low turbulence intensities, transition follows natural mechanisms described by linear stability theory (Tollmien (1928)). As turbulence intensity increases, instabilities grow until the bypass transition occurs (Morkovin (1993)), where classical Tollmien-Schlichting (TS) waves and secondary structures are bypassed, and turbulence patches form directly. Between these extremes, transient growth occurs, where natural and bypass transition mechanisms coexist. In this phase, Tollmien-Schlichting waves still emerge at low turbulence levels but transition to secondary instabilities more rapidly. Receptivity (Saric et al. (2002)) describes how boundary layers absorb and amplify external disturbances, influenced by turbulence intensity, surface roughness, and acoustic excitation. According to Saric et al. (2002) and Mayle (1991), the role of receptivity in boundary layers is of utmost importance for the transient growth. Since this study focuses solely on free-stream turbulence, only this aspect is further discussed in Sec. 2.6.2.

Surface Roughness

Surface roughness introduces additional disturbances. However, if the roughness elements

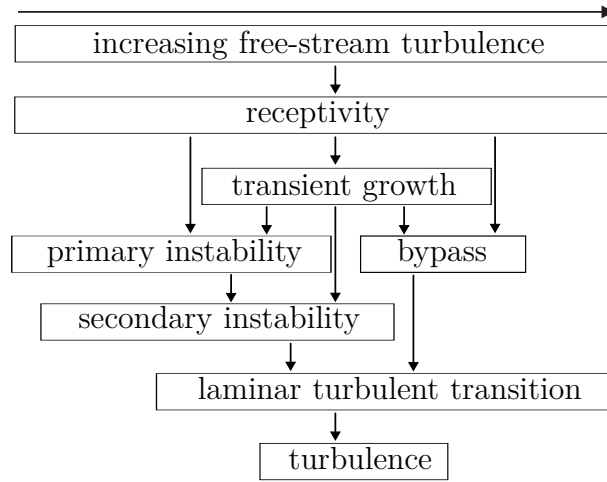


Figure 2.8: The path from receptivity to turbulence after Saric et al. (2002)

are confined within the viscous sublayer, they have little impact on transition. However, these are not relevant for the flow conditions considered in this work.

Compressibility

In supersonic flows, additional disturbances such as Mack modes can occur. However, these are not relevant for the flow conditions considered in this work.

In summary, the formation and amplification of disturbances, and thus the onset of transition, depend on a complex interplay between the aforementioned factors. A comprehensive grasp of these mechanisms enables better transition modeling and control in turbomachinery design. In this work, the influence of free-stream turbulence on transition is investigated, while also acknowledging that the pressure gradient is an inherent factor in turbine and compressor flows, affecting the transition.

2.6.1 Transition Types

Natural Transition

Natural transition occurs in wall-bounded flows subjected to minimal external disturbances, typically when free-stream turbulence levels remain below 0.5%. Figure 2.9 illustrates this process using a flat plate boundary layer at zero incidence. Initially, disturbances within the laminar boundary layer arise due to Tollmien-Schlichting waves, which represent the primary instabilities. As these waves amplify, they lead to the emergence of secondary three-dimensional instabilities, forming complex flow structures such as Λ -shaped vortices in both the streamwise and cross-flow directions (non-linear interactions stage in Fig. 2.9). In the final phase, these large-scale vortices break down into turbulent spots, which merge to form a fully turbulent boundary layer. According to Yang et al. (2001), natural transition is the most extensively researched transition type. Its process is relatively well understood, with significant contributions from linear stability theory and experimental investigations. Marsden et al. (2001) and Yang et al. (2001) outline five distinct phases of natural transition.

1. Ambient stochastic perturbations are converted into growing eigenmodes, initiating disturbances within the boundary layer.

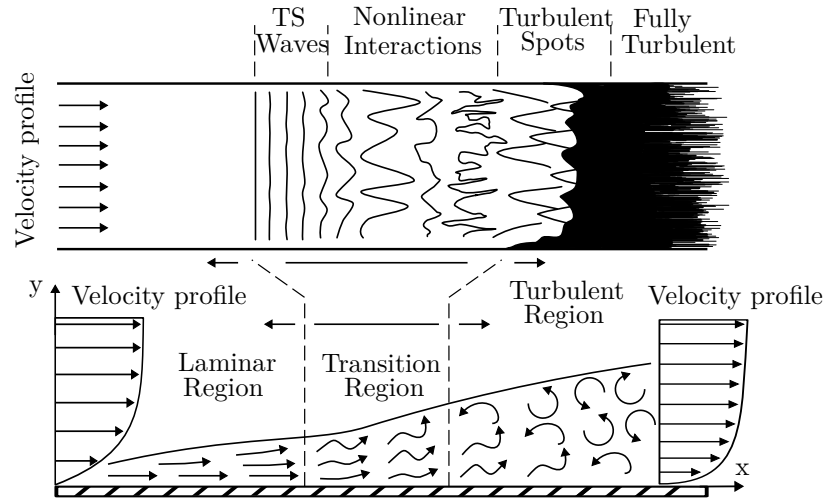


Figure 2.9: Sketch of natural transition of boundary layer for a flat plate at zero incidence

2. Small disturbances, particularly two-dimensional Tollmien-Schlichting waves, are linearly amplified until they reach amplitudes where nonlinear effects become significant.
3. As disturbances reach a finite amplitude, they either saturate and transform the flow into a new, steady state or directly trigger a bypass transition, skipping the gradual growth phases.
4. In secondary instability phase, the new steady or quasi-steady flow acts as the base flow for secondary instabilities, which develop into complex three-dimensional flow structures. This stage typically evolves more rapidly than the primary instability phase.
5. The final stage is characterized by the rapid onset of non-linearities and higher-order instabilities (Breakdown stage), such as symmetry-breaking modes, leading to a cascade of scales and frequencies within the flow. This breakdown process is highly complex and is often best analyzed using numerical experiments.

This five-phase framework provides a structured description of natural transition, highlighting the interplay between linear and nonlinear mechanisms driving the laminar-to-turbulent process. However, in practical applications, clear boundaries between these stages may not always be distinguishable due to flow complexities. Nevertheless, this classification remains a valuable tool for understanding and predicting transition behavior in boundary layers.

Bypass Transition

Bypass transition occurs when the free-stream turbulence intensity exceeds approximately 1 %, significantly accelerating the transition process and effectively circumventing the classical two-dimensional Tollmien-Schlichting (T-S) instability mechanism associated with natural transition. In the framework of linear stability theory, T-S waves arise as modal eigen-solutions of the Orr-Sommerfeld equation, which governs the temporal or spatial evolution of infinitesimal disturbances in parallel shear flows under the assumption of linearized Navier-Stokes dynamics. In contrast, bypass transition is dominated by non-modal disturbance growth mechanisms, where transient energy amplification leads to the

formation of streamwise-elongated vortex and streak structures, ultimately triggering turbulence without the amplification of classical T–S waves. First introduced by Morkovin (1993), bypass transition arises when the exponential growth of Tollmien-Schlichting waves is replaced by alternative amplifying processes. This process manifests through elongated backward jets in the fluctuating streamwise velocity field, moving opposite to the local mean flow. These backward jets extend into the upper part of the boundary layer, where they interact with free-stream eddies. In certain regions, perturbations in the jet’s shear layer evolve into localized turbulent spots, patches of irregular motion propagating longitudinally and laterally. These turbulent spots eventually merge to form a fully developed turbulent boundary layer, defining the upstream boundary of turbulence.

Another interpretation of bypass transition focuses on the effect of free-stream turbulence on the boundary layer. With increasing turbulence intensity, the boundary layer experiences low-frequency disturbances penetrating from the free-stream, undergoing algebraic (or transient, non-modal) growth. A key feature of bypass transition is the formation of Klebanoff streaks, regions of alternating high- and low-speed fluid, originally described by Klebanoff et al. (1962) as periodic thickening and thinning of the boundary layer. According to Jacobs et al. (1998), the boundary layer acts as a low-pass filter through a process called shear-sheltering, allowing only low-frequency perturbations to penetrate. These disturbances elongate under shear forces, forming long, streamwise streaks (cf. Fig. 2.10 bottom), while the free-stream remains relatively isotropic. Low-speed Klebanoff streaks can detach from the wall, lifting low-momentum fluid upward in a manner similar to backward jets. Higher-frequency free-stream turbulence mainly affects the upper boundary layer, destabilizing backward jets and triggering local breakdown into turbulence (as seen in Fig. 2.10 middle).

As Klebanoff streaks travel downstream, their amplitude increases, making them prone to additional instabilities. The transition process reaches its peak at the outer boundary layer edge, where interactions between high-frequency free-stream disturbances and low-velocity fluid within streaks lead directly to the formation of turbulent spots. These spots merge, completing the transition to a fully turbulent boundary layer.

Bypass transition represents an alternative route to turbulence that differs fundamentally from Tollmien-Schlichting-wave-driven natural transition. It underscores the complex interplay between free-stream turbulence and boundary layer dynamics, revealing a direct pathway to turbulence through streak formation, jet interactions, and turbulent spot generation.

Wake induced Transition

Wake-induced transition occurs when periodic disturbances, such as wakes shed from upstream blades in turbomachinery, interact with the boundary layer, accelerating the transition to turbulence. Depending on the Reynolds number and flow conditions, this process can manifest in attached boundary layers, separated shear layers, or in the presence of laminar separation bubbles. At high Reynolds numbers, wakes can trigger transition without prior separation, while at lower Reynolds numbers, they can destabilize separated shear layers, leading to an earlier onset of turbulence. The frequency and strength of these wake interactions play a crucial role in determining the transition pathway (Halstead et al. (1997)). However, the wake-induced transition is not the scope of this work and will not be discussed further.

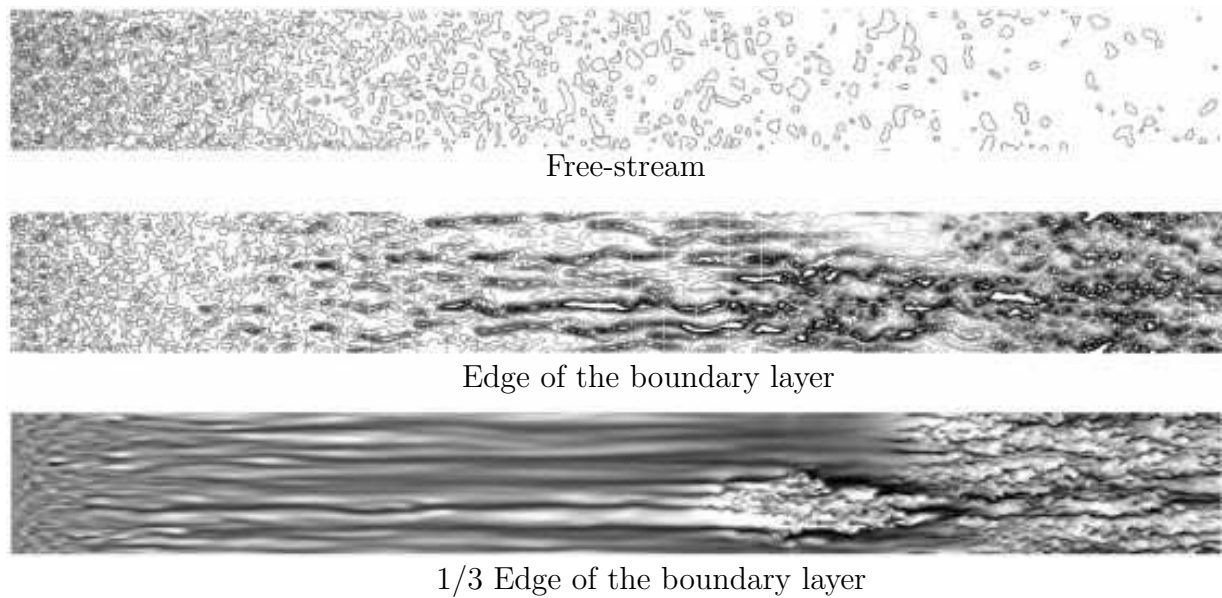


Figure 2.10: Concept of shear shelter based on the streamwise fluctuating velocity in free-stream, at boundary layer edge and close to the wall (reproduced from Jacobs et al. (1998))

Transition in laminar separation bubbles

Laminar separation bubbles (LSBs) are a common phenomenon in aerodynamic flows, particularly in low-pressure turbine (LPT) stages operating at low Reynolds numbers. They form when an adverse pressure gradient causes the laminar boundary layer to detach from the surface, creating a region of separated flow. As shown in Fig. 2.11, this separated region consists of two primary zones. A recirculating region beneath the mean dividing streamline, containing slow-moving fluid (1), and a free shear layer above, where velocity gradients lead to instabilities and eventual transition to turbulence (2).

Upstream of the transition point, the free shear layer remains laminar due to insufficient shear stress to counteract the adverse pressure gradient, resulting in a nearly uniform pressure distribution (shown in top row of Fig. 2.11). However, once transition occurs, the turbulent entrainment process significantly enhances momentum transfer, leading to a pressure recovery and eventual flow reattachment, forming a closed separation bubble (cf. Fig. 2.11). The pressure rise within the bubble, often described as a near-linear increase post-transition, is driven by the expansion of the shear layer and is different than the geometric separation over sharp corners. A key instability mechanism in LSBs is the Kelvin-Helmholtz (KH) instability, which arises due to the inflectional velocity profile of the separated shear layer. This leads to the formation of coherent spanwise vortices that break down into turbulence through secondary instabilities, marking the transition point near the maximum height of the bubble. If reattachment occurs downstream, the bubble is classified as a short separation bubble, shown in bottom row of Fig. 2.11, (Horton (1968)), typically causing minimal impact on the overall pressure distribution. In contrast, a long separation bubble exhibits an extended pressure plateau and may result from the bursting of a short bubble. The nature of transition in separated shear layers is influenced by boundary conditions, particularly Reynolds number and free-stream turbulence intensity (Walker (1993)).

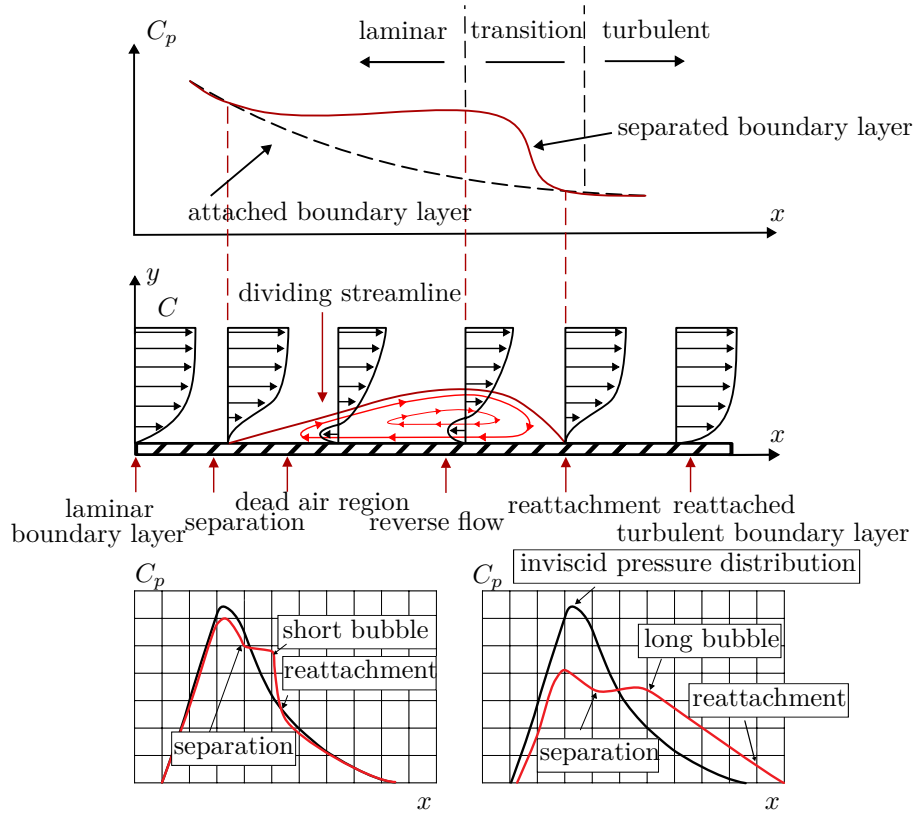


Figure 2.11: Top: Flow structure, in a time-averaged sense, of a laminar separation bubble and the corresponding pressure distribution adapted from Himmel (2010). Bottom: Effect of short (left) and long (right) laminar separation bubble on pressure distribution adapted from Gaster (1969).

At low turbulence intensities and Reynolds numbers, the flow remains laminar up to the trailing edge, transitioning only in the wake. With increasing Reynolds number, separation-induced transition occurs, while at sufficiently high free-stream turbulence intensities, bypass transition can suppress the bubble entirely. This highlights the strong dependence of LSB dynamics on external disturbances, making them a critical factor in turbomachinery performance.

In this work, particular attention is given to the formation and transition mechanisms of laminar separation bubbles, especially in LPT blades, where they play a key role in aerodynamic efficiency and flow loss generation.

2.6.2 Effect of free-stream turbulence on transition

The influence of free-stream disturbances, such as free-stream turbulence intensity, on the receptivity of the boundary layer was briefly discussed in Sec. 2.6. In the current section, the influence of both low and elevated free-stream turbulence conditions on transition is examined, with particular emphasis on the laminar separation bubble.

Transition at low free-stream turbulence intensity

The transition mechanism within a laminar separation bubble (LSB) is strongly influenced by free-stream turbulence intensity. In low free-stream turbulence conditions, transition within a laminar separation bubble is dominated by the growth and breakdown of Kelvin-Helmholtz (KH) instabilities. These initially two-dimensional rollers evolve into three-dimensional turbulent structures after undergoing secondary instabilities (as described in Sec. 2.6). The amplification rate of these disturbances is significantly higher than that of Tollmien-Schlichting (TS) waves, which typically dominate transition in attached boundary layers.

Figure 2.12 illustrates this transition process, showing the evolution of the KH instability and the breakdown to turbulence. In particular, Fig. 2.13 highlights the evolution of maximum turbulent kinetic energy ($\text{TKE}_{\max}$), while varying the free-stream turbulence intensity, along the suction side of the LPT blade, demonstrating the saturation of turbulence as it reaches a peak level before decaying. Although Tollmien-Schlichting waves may interact with KH instabilities at low turbulence intensities, their influence on turbulence breakdown remains debated. Unlike transition in attached boundary layers, where the secondary instability mechanisms are well understood, transition in separated flows remains less explored. Literature suggests that the reverse flow region within the separation bubble, as shown in Fig. 2.11, plays a crucial role in secondary amplification. This region generates localized high-shear zones that enhance three-dimensional perturbations, which are then transported downstream by large-scale two-dimensional motions. When these three-dimensional modes reach sufficient amplitudes, they disrupt the shear layer, triggering full transition to turbulence and leading to the formation of spanwise turbulent rolls.

Conceptually, the LSB consists of two distinct instability-driven regions. The outer Shear Layer Region, governed by inviscid instability, where KH vortices dominate and are influenced by Reynolds number effects. Near-Wall Region, controlled by viscous instability, where disturbances grow at lower frequencies due to the presence of strong velocity gradients. The location of the inflection point in the velocity profile determines which instability mode dominates. At higher frequencies, the outer inviscid instability prevails, while at lower frequencies, the near-wall viscous instability becomes significant. Understanding the interaction of these mechanisms remains a key challenge in predicting transition in separated flows. Such interactions between different frequencies can be effectively examined using spectral analysis, as demonstrated in Sec. 5.2.3 and Sec. 6.2.3.

Transition at elevated free-stream turbulence intensity

In realistic turbomachinery applications, the free-stream turbulence intensity (FSTI) is significantly higher than in low-disturbance environments, reaching up to 8 % in low-pressure turbines (LPTs) and 10 % in high-pressure compressors (HPCs). Elevated turbulence levels introduce non-modal low-frequency disturbances (Klebanoff streaks, cf. Sec. 2.6.1), which significantly alter the transition process in LSBs. At moderate turbulence levels, Klebanoff streaks interact with the detached shear layer, modifying the classical Kelvin-Helmholtz (KH) instability-driven transition process. KH-rollers are distorted by three-dimensional Klebanoff modes, leading to an accelerated breakdown into small-scale turbulence. This interaction results in faster transition and a shorter separation bubble compared to low-FST cases. Studies suggest that Klebanoff streaks primarily influence the upstream portion of the boundary layer rather than the turbulent reattachment region (Vadlamani (2014)). At higher turbulence intensities, Klebanoff streaks play a dominant role in promoting early

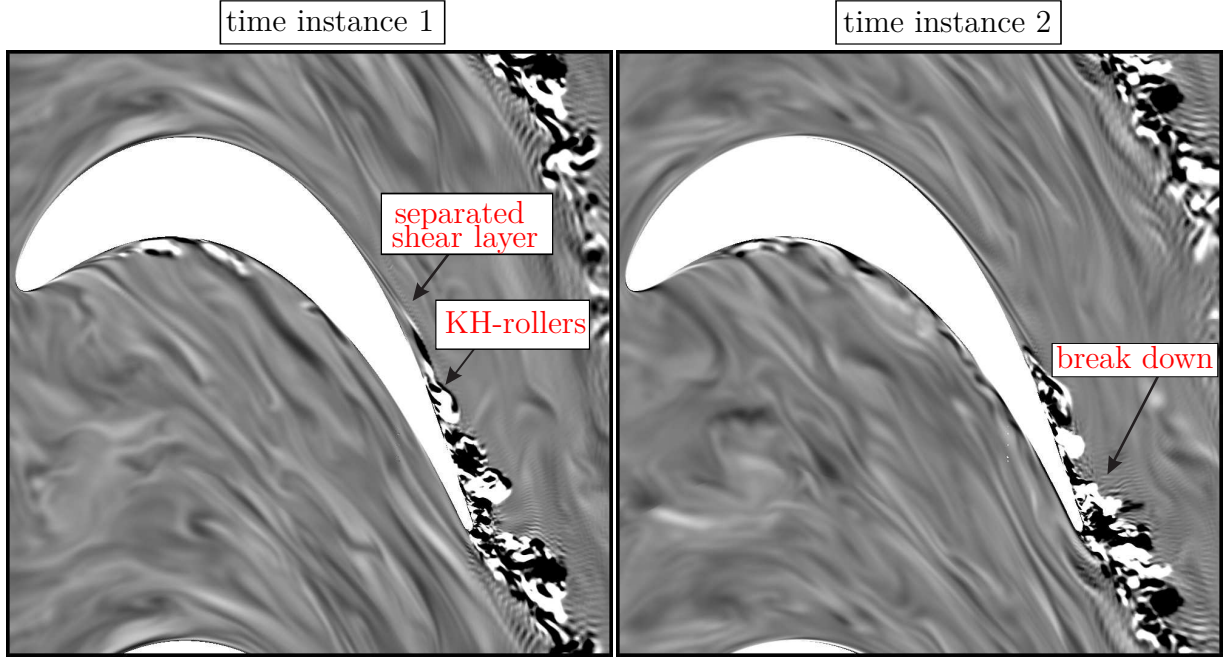


Figure 2.12: Illustration of the KH shear layer roll-up and breakdown mechanism at mid-span flow of MTU-T161 for $Re=90,000$ by LES at a moderate free-stream turbulence level (10 %), calculated in the current work.

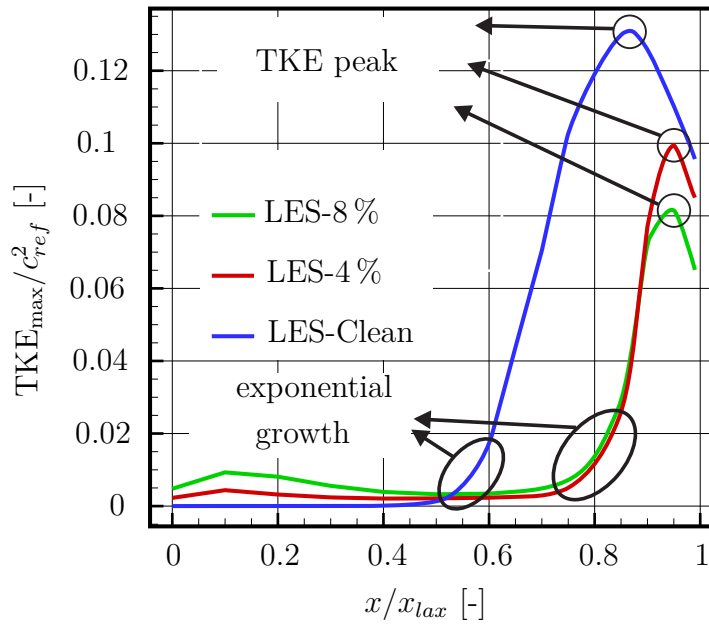


Figure 2.13: Evolution of maximum TKE along the suction side flow of MTU-T161 flow for $Re=90,000$ by LES for three free-stream turbulence levels, calculated in the current work.

breakdown into three-dimensional turbulence. Although elevated FST accelerates bypass transition, KH instabilities remain active and contribute to the breakdown process. Their interaction with non-modal Klebanoff streaks results in a more direct and faster transition (Vadlamani (2014)).

2.7 Flow Losses

Turbomachinery flow losses can be classified into various categories depending on the component under investigation. For low-pressure turbines, these flow losses are typically grouped as follows

1. The profile flow losses encompass all energy losses related to boundary layer phenomena, such as flow separation and viscous dissipation. Profile flow losses are the dominant mechanism, contributing between 40 % and 60 % of the total flow losses.
2. The side-wall flow and side-wall boundary layer losses arise from the formation of vortices caused by pressure gradients along the boundary layers at the hub and shroud, such as passage and horseshoe vortices. Additionally, interactions between the blade and the side-walls generate secondary flows. Side-wall flow losses can account for up to 25 % of the total flow losses.
3. The tip leakage flow losses occur due to the formation of vortical structures and mixing zones in the clearance between the blade tip and the casing. Tip leakage flow losses contribute up to 10 % of the total flow losses.
4. The wakes shed by upstream blades mix with the downstream flow, resulting in additional flow losses that typically range between 5 % and 10 %, for low and mid subsonic flow velocities while increasing at higher velocities. These losses are called wake mixing flow losses.
5. In low-pressure turbines operating at low Reynolds numbers, laminar separation followed by turbulent reattachment can occur, particularly on the suction side. This process can constitute a significant contribution to the overall aerodynamic losses in such operating regimes, commonly referred to as laminar separation and reattachment losses.

A similar classification can be applied to compressor cascades, with some minor differences. In compressor cascades, the effect of the laminar separation bubble is typically less pronounced (for high Reynolds numbers or high Mach numbers), while flow losses associated with shock and unsteady flow, common in transonic compressors, may be observed. Moreover, the estimated contributions of each flow loss mechanism differ between low-pressure turbines and high-pressure compressors. In both cases, profile flow losses constitute the largest share of the overall efficiency loss. Notably, in low pressure turbine cascades, profile flow losses tend to be more significant than in compressor cascades due to the typically high aspect ratio of low-pressure turbine blades (according to Curtis et al. (1997), 60 % associated to suction side flow losses and 20 % to pressure side surface). In this work, the focus is placed on profile flow losses, despite the fact that both geometries are calculated as three-dimensional models including the side-walls. Nevertheless, as stated in the literature, minimizing the profile flow losses plays a key role in enhancing overall turbomachinery efficiency.

Several analyses can be employed to characterize flow losses in turbomachinery, with the most commonly used flow loss coefficients being those related to entropy (s), enthalpy (h), and total pressure. According to Denton (2010), the entropy and enthalpy loss coefficients yield identical results. However, the entropy loss coefficient offers a distinct advantage in engine applications. Jardine (2019) provides a detailed examination of each flow loss

analysis method and its limitations. Although there is general consensus that the entropy loss coefficient is the most accurate for evaluating engine performance, industrial practice often favors the total pressure loss coefficient because it is relatively easier to measure in experimental test rigs. In the current work, the total pressure loss equation (cf. (2.51)), which is widely used to quantify flow losses in turbomachinery blades and to compare numerical results with experimental measurements, is used to quantify the flow losses. Over time, the growing capabilities of accurate numerical methods have led to an increased use of alternative flow loss prediction techniques. For each of the methods (entropy, enthalpy, and total pressure), a corresponding flow loss coefficient can be defined. For example, by taking time-averaged and flux-averaged entropy values at the inlet (indicated as 1) and the outlet (indicated as 2) of the domain the entropy loss coefficient

$$\zeta^s = \frac{T_2(s_2 - s_1)}{h_{t,1} - h_1}, \quad (2.91)$$

can be computed. The entropy can be related to time-averaged and mass-averaged pressure and temperature as follows

$$s - s_{\text{ref}} = C_p \ln \left(\frac{T}{T_{\text{ref}}} \right) - R \ln \left(\frac{p}{p_{\text{ref}}} \right), \quad (2.92)$$

where R is the gas constant and C_p the heat capacity. The enthalpy loss coefficient, in a time averaged and mass averaged sense, can be computed as follows

$$\zeta^h = \frac{h_2 - h_{2s}}{h_{t,1} - h_1}, \quad (2.93)$$

where h_{2s} is the isentropic downstream stagnation enthalpy (calculated assuming no losses). One of the most widely used methods for quantifying flow losses in turbomachinery is the calculation of the total pressure loss coefficient, which is determined experimentally by directly measuring the total pressure

$$\zeta^p = \frac{p_{t,1} - p_{t,2}}{p_{t,1} - p_1}. \quad (2.94)$$

Flow loss coefficients offer an estimate of the overall performance loss in a system, while numerical studies, particularly those using high-fidelity methods, yield detailed insights into the underlying flow loss mechanisms. In this context, examining the transport equations provides a more comprehensive understanding of these flow losses.

One approach, as described by Zhao et al. (2020b), is to assess the entropy generation within the computational domain. To achieve this, the entropy transport equations, based on averaged flow quantities, are derived. According to the second law of thermodynamics, which states that the total entropy S of a system can be expressed as follows

$$\dot{S} + I_s = \int_V G_s dV \geq 0, \quad (2.95)$$

where $\dot{S}$ is the time-rate-of-change of the entropy in the system, I_s is the entropy exchange with environment, and the right-hand side term represents the entropy generation rate G_s

integrated over the computational volume V (Camberos (2001), Zhao et al. (2020b)). The entropy balance equation can be written as following

$$\frac{\partial(\rho s)}{\partial t} + \frac{\partial F_i}{\partial x_i} \equiv G_s \geq 0. \quad (2.96)$$

In (2.96), ρ is density, s is the specific entropy, and F_i is the entropy flux caused by mass flux and heat transfer (Adeyinka et al. (2004)). In the next step, the first law of thermodynamics is used

$$de = Tds - pd\frac{1}{\rho}, \quad (2.97)$$

and the equation for the internal energy $e = e_0 - 1/2c_i c_i$

$$\frac{\partial(\rho e)}{\partial t} + \frac{\partial(\rho c_j e)}{\partial x_j} + p \frac{\partial c_j}{\partial x_j} = \sigma_{ij} \frac{\partial c_i}{\partial x_j} - \frac{\partial q_j}{\partial x_j} \quad (2.98)$$

Combining (2.97) and (2.98) the entropy balance equation can be derived

$$\frac{\partial(\rho s)}{\partial t} + \frac{\partial(\rho c_i s)}{\partial x_i} + \frac{\partial}{\partial x_i} \left(\frac{q_i}{T} \right) = \frac{1}{T} \sigma_{ij} \frac{\partial c_j}{\partial x_i} + \frac{c_p \mu}{\text{Pr}} \frac{1}{T^2} \left(\frac{\partial T}{\partial x_i} \right)^2. \quad (2.99)$$

Based on (2.95), (2.96), (2.97), (2.98) and (2.99) it can be concluded that the entropy generation mechanisms is related to heat transfer and viscous dissipation as following

$$G_s^{NS} = \underbrace{\frac{1}{T} \sigma_{ij} \frac{\partial c_j}{\partial x_i}}_{\text{vis. dissipation}} + \underbrace{\frac{c_p \mu}{\text{Pr}} \frac{1}{T^2} \left(\frac{\partial T}{\partial x_i} \right)^2}_{\text{heat transfer}}. \quad (2.100)$$

(2.100) is the local entropy generation in the instantaneous flow fields described by Navier-Stokes equations. By taking the average of the instantaneous entropy generation equation (2.100), the contribution of the mean flow viscous dissipation and turbulent dissipation can be distinguished. Calculating the entropy generation based on mean flow fields can be written as following

$$\begin{aligned} \frac{\partial(\bar{\rho} \tilde{c}_i \tilde{s})}{\partial x_i} + \frac{\partial}{\partial x_i} \left(\frac{\tilde{q}_i}{\tilde{T}} \right) &= \frac{1}{\tilde{T}} \sigma_{ij}(\tilde{c}) \frac{\partial \tilde{c}_j}{\partial x_i} + \frac{c_p \mu}{\text{Pr}} \frac{1}{\tilde{T}^2} \left(\frac{\partial \tilde{T}}{\partial x_i} \right)^2 \\ &+ \frac{1}{\tilde{T}} \tau_{ij}^R \frac{\partial \tilde{c}_j}{\partial x_i} - \frac{1}{\tilde{T}} \frac{\partial}{\partial x_i} (q_i^R) - \frac{1}{\tilde{T}} \frac{\partial(\bar{\rho} \tilde{c}_i k)}{\partial x_i}, \end{aligned} \quad (2.101)$$

where, $\bar{\phi}$ is the time-average of a random variable, ϕ , and $\tilde{\phi}$ represents a Favre-averaged quantity. The left-hand side terms of the (2.101) represents the entropy variations caused by mass and heat fluxes. The entropy balance equation for RANS flow fields can be derived

as following

$$\begin{aligned}\tilde{G}_s^{\text{RANS}} = & \underbrace{\frac{1}{\tilde{T}}\sigma_{ij}(\tilde{c})\frac{\partial\tilde{c}_j}{\partial x_i}}_{V_M} + \underbrace{\frac{c_p\mu}{\text{Pr}}\frac{1}{\tilde{T}^2}\left(\frac{\partial\tilde{T}}{\partial x_i}\right)^2}_{H_M} \\ & + \underbrace{\frac{1}{\tilde{T}}\tau_{ij}^R\frac{\partial\tilde{c}_j}{\partial x_i}}_{P_R} - \underbrace{\frac{1}{\tilde{T}}\frac{\partial}{\partial x_i}\left(q_i^R\right)}_{H_R} - \underbrace{\frac{1}{\tilde{T}}\frac{\partial(\bar{\rho}\tilde{c}_ik)}{\partial x_i}}_{A_{TKE}}.\end{aligned}\quad (2.102)$$

In the above equation, the term V_M , represents the entropy generation caused by the mean viscous dissipation, H_M the mean flow irreversible heat flux, P_R the turbulence production related to the Reynolds stress, H_R the turbulent heat flux and A_{TKE} the advection of TKE. The entropy balance equation for Favre-averaged LES flow fields is written in a similar manner as following

$$\begin{aligned}\tilde{G}_s^{\text{LES}} = & \underbrace{\frac{1}{\tilde{T}}\sigma_{ij}(\tilde{c})\frac{\partial\tilde{c}_j}{\partial x_i}}_{V_M} + \underbrace{\frac{c_p\mu}{\text{Pr}}\frac{1}{\tilde{T}^2}\left(\frac{\partial\tilde{T}}{\partial x_i}\right)^2}_{H_M} + \underbrace{\frac{1}{\tilde{T}}\tau_{ij}^R\frac{\partial\tilde{c}_j}{\partial x_i}}_{P_R} - \underbrace{\frac{1}{\tilde{T}}\frac{\partial}{\partial x_i}\left(q_i^R\right)}_{H_R} - \underbrace{\frac{1}{\tilde{T}}\frac{\partial(\bar{\rho}\tilde{c}_ik)}{\partial x_i}}_{A_{TKE}} \\ & + \underbrace{\frac{1}{\tilde{T}}\tau_{ij}^{\text{SGS}} + \frac{\partial\tilde{c}_j}{\partial x_i}}_{V_S} - \underbrace{\frac{1}{\tilde{T}}\frac{\partial}{\partial x_i}\left(\tilde{q}_i^{\text{SGS}}\right)}_{H_S}\end{aligned}\quad (2.103)$$

Using (2.103), flow losses are characterized and related to entropy generation, and the various contributing terms and mechanisms are distinguished.

2.7.1 Turbine Flow - Transition and Flow Losses

The Reynolds numbers in the low-pressure turbine section of an engine are the lowest among different components due to the relatively low blade circumferential speeds, low density, low Mach numbers of the flow and small reference chord length. Additionally, in highly loaded turbine blades, the flow undergoes significant turning to achieve the required energy conversion, which further influences the isentropic Mach number distribution. The aforementioned conditions result in extensive laminar boundary layer regions on the suction side of the blade profiles. The presence of an adverse pressure gradient makes these laminar boundary layers highly unstable and prone to separation. The length of the separation bubble (cf. Fig. 2.11) plays a crucial role in determining profile flow losses and overall turbine efficiency. Designing blade profiles is particularly challenging, as modern aircraft engines operate across a wide range of conditions. In low-pressure turbines, the Reynolds number typically ranges from approximately 100,000, during cruise to around 400,000 during takeoff. Figure 2.14 illustrates the relationship between the flow loss coefficient and Reynolds number for different flow scenarios. The blue-shaded regions in the diagram indicate the friction losses within the shear layers, while the red-highlighted areas represent the additional flow losses caused by flow separation.

As shown in numerous studies, laminar boundary layers generally cause lower friction losses than turbulent boundary layers. However, laminar boundary layers are also more prone to flow separation. At very low Reynolds numbers (left side of Fig. 2.14), an open separation

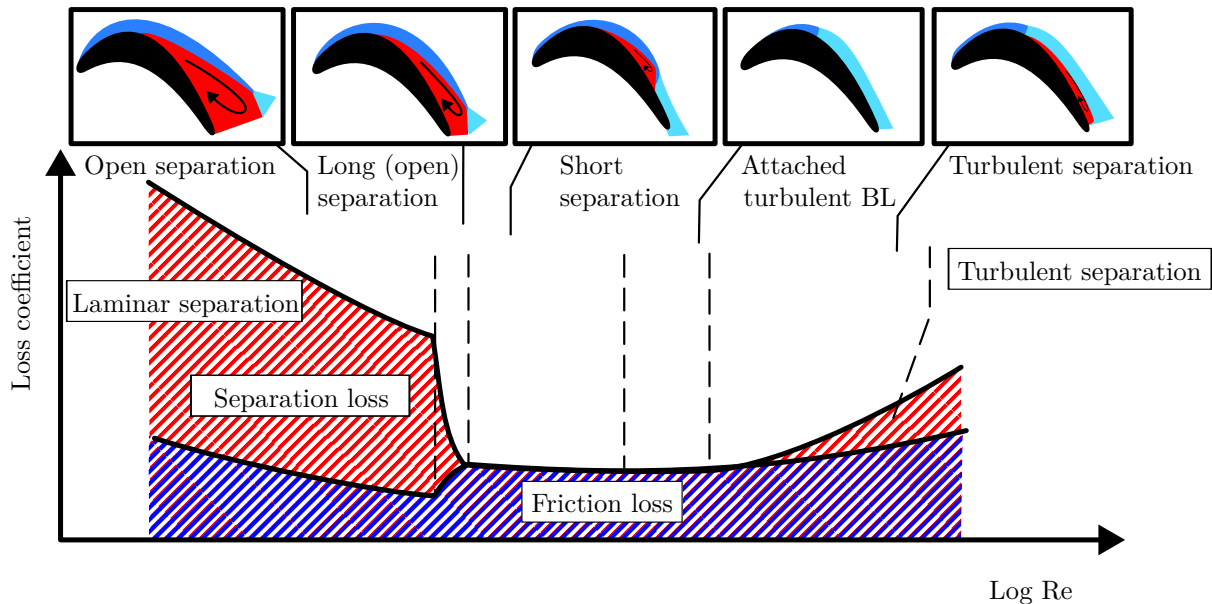


Figure 2.14: Flow losses over Reynolds number for a low-pressure turbine. Figure adapted from Kožulović (2008) and Hourmouziadis (1989)

bubble leads to increased separation losses. If the flow reattaches and transitions to turbulence within a laminar separation bubble (middle region of Fig. 2.14), separation losses are eliminated, significantly reducing total flow losses. As the Reynolds number increases, transition occurs within the attached boundary layer via bypass transition. This scenario, more common in low-pressure turbines during takeoff, results in relatively low flow losses. Under stronger adverse pressure gradients, the flow can undergo turbulent separation. In this case, the flow losses are considerably lower compared to laminar flow separation, as turbulent separation occurs later, allowing for greater pressure recovery before the flow detaches. In summary, avoiding open separation bubbles, whether laminar or turbulent, is critical to maintaining high low-pressure turbine efficiency. For highly loaded profiles, where the risk of separation is elevated, a thorough understanding of boundary layer development along the suction side of turbine blades is essential. This includes the growth of the laminar boundary layer and its transition to turbulence, whether through bypass transition or via a detached shear layer forming a laminar separation bubble.

2.7.2 Compressor Flow - Transition and Flow Losses

Despite the high Reynolds numbers and free-stream turbulence intensities (FSTI) typical of gas turbine compressors, large portions of the boundary layer remain in a transitional state. Experimental studies by Steinert et al. (1996) demonstrated that even at a Reynolds number of 840,000 and an FSTI of 2.5% the suction-side boundary layer remained laminar up to the peak suction point across a wide range of incidence angles (Przytarski et al. (2020)). Similarly, Schreiber et al. (2004) showed that laminar separation in outlet guide vane flows can significantly increase profile flow losses in small turbofan engines, with a critical Reynolds number of approximately 200,000. Although this value is lower than those in conventional aircraft engines (typically between 500,000 and 1,500,000), it underscores

the growing importance of transition effects in smaller engine designs (Przytarski et al. (2020)).

Compressor transition is a complex process involving both laminar-to-turbulent (forward) and turbulent-to-laminar (reverse) transitions, as discussed by Mayle (1991). Among the key influencing factors, free-stream turbulence plays a dominant role. Direct numerical simulations (DNS) by Zaki et al. (2010) revealed that at low turbulence intensities, transition is primarily separation-induced on both suction and pressure surfaces. At a moderate FSTI of 3.25%, transition on the pressure surface shifts from purely bypass-type behavior to a form more closely resembling natural transition. At turbulence levels above 6%, classical bypass transition dominates, and by 10%, flow separation is entirely suppressed on both surfaces. Separated-flow transition, although critical to compressor performance, has received relatively little attention. Since compressor airfoils often experience some degree of separation, especially at off-design conditions, their performance is highly sensitive to the characteristics of the separated flow. Incorporating controlled transition via separation bubbles into the design process could therefore yield significant efficiency gains.

Compressor flows are inherently complex, involving three-dimensional side-wall effects and clearance-leakage flows. Nevertheless, blade skin-friction alone accounts for approximately 40% of stage efficiency losses under design conditions (Dunham (1991)), with flow losses increasing substantially under off-design operation due to large-scale separation. In addition to free-stream turbulence and pressure gradients, the leading-edge geometry significantly affects transition behavior. However, this aspect lies beyond the scope of the present study. As in low-pressure turbines, transition and separation are closely coupled in compressors, and understanding their interaction is essential. Effective transition management can minimize flow losses under design conditions and improve off-design performance, thereby expanding the operational envelope. Given the diverse range of transition phenomena in compressors, including adverse pressure gradients, separated flows, and wake induced transition, a robust understanding of these mechanisms is vital for further efficiency improvements.

Figure 2.15 illustrates the breakdown of flow loss mechanisms (as entropy production rate) in a compressor cascade across a range of inlet inflow incidence angles at a specified Mach number. In this context, positive incidence corresponds to the inflow being directed toward the pressure side (PS) of the blade, while negative incidence indicates flow directed toward the suction side (SS). The figure uses color bands to represent individual flow loss mechanisms, with the vertical thickness of each colored region indicating the magnitude of that specific flow loss contribution relative to the total flow loss. At negative incidence angles, a shock terminates a small supersonic patch near the leading edge of the pressure surface. This interaction induces boundary layer separation, but the associated flow losses primarily arise from mixing within the separation bubble, rather than from the shock itself. This regime is affected by both passage-scale effects, such as the reduction in geometric throat area, and local flow features, like the formation of the supersonic patch at the leading edge.

The dominant flow loss mechanism at large negative incidence is the leading edge (LE) separation on the pressure side, as indicated by the red region in the figure. At design incidence (around 0°), the flow remains largely attached, and the overall profile flow loss is at a minimum. In this range, the most significant contributions come from attached flow on the suction side (gray area), along with relatively small flow losses due to attached flow on

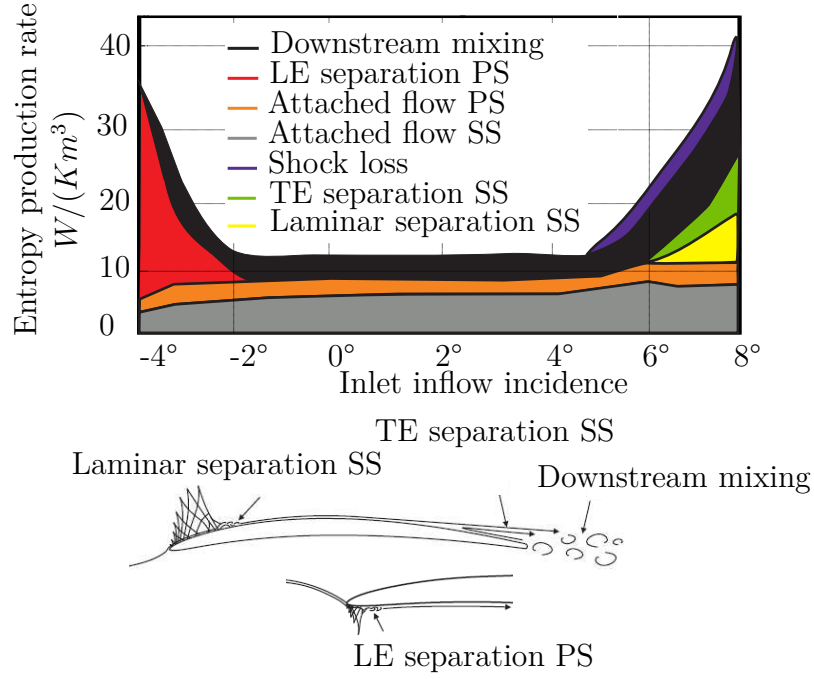


Figure 2.15: Breakdown of flow loss mechanism over a compressor cascade with varying incidences at $M = 0.77$, $Re=660,000$ from Goodhand et al. (2012)

the pressure side (orange area) and downstream wake mixing (black area). The dominance of these mechanisms indicates an efficient operating point with minimal flow separation. At positive incidence, another shock terminates a supersonic patch, this time on the suction surface, and causes laminar boundary layer separation. The resulting increase in flow losses is attributed to three mechanisms. Mixing in the reattaching laminar separation bubble (yellow), trailing edge (TE) separation on the suction surface due to boundary layer thickening (green), and downstream wake mixing (black), which increases as the boundary layer fails to withstand the adverse pressure gradient. As positive incidence increases beyond 4° , these mechanisms become more pronounced. By 8° , downstream mixing, laminar separation on the suction side, and TE separation are clearly the dominant contributors to profile flow loss.

Another factor that significantly influences the boundary-layer development in compressor cascades is the non-equilibrium behavior of turbulence, which manifests as a temporal and spatial lag between turbulence production and dissipation. This non-equilibrium state has a pronounced impact on the associated loss mechanisms. A schematic illustration of such non-equilibrium behavior within the boundary layer of a compressor cascade is provided in Fig. 2.16. Although most conventional turbulence models assume a local equilibrium between production and dissipation, experimental and numerical studies in the literature indicate that up to 30 % of the suction-side boundary layer can exhibit pronounced non-equilibrium characteristics.

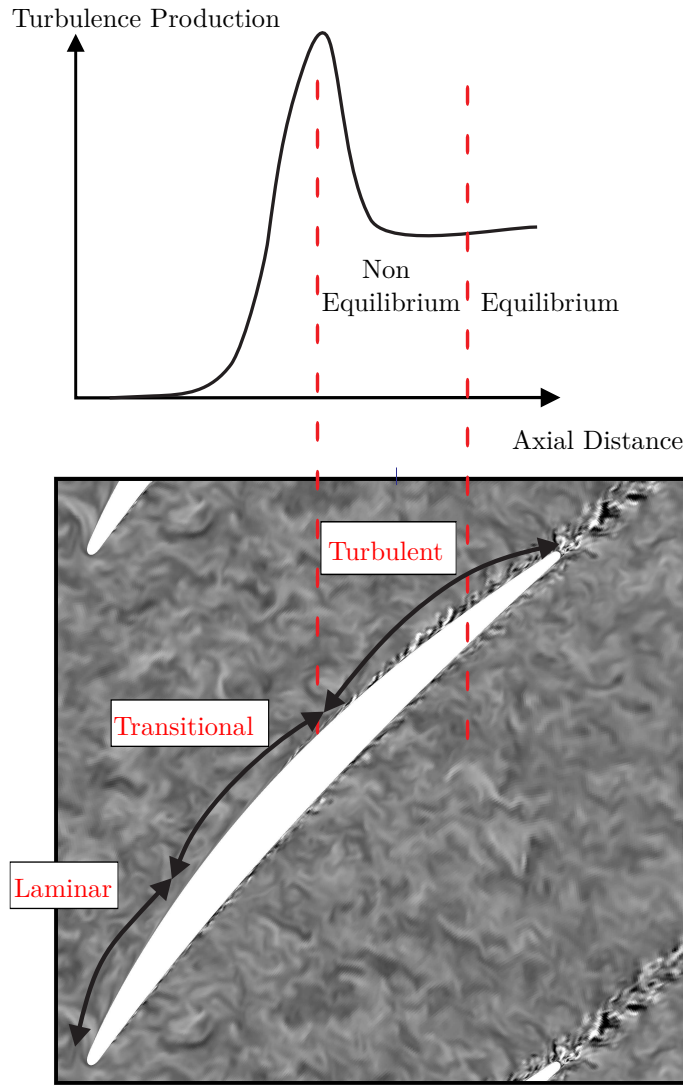


Figure 2.16: Top: Evolution of turbulence production, equilibrium states and schematic of boundary layer states on a compressor blade suction surface adapted from Wheeler et al. (2018). Bottom: Boundary layer states over the suction side of the HPC cascade used in the current work.

2.8 Post Processing Methods

In the current section, the post processing methods, used in this work, are briefly introduced.

2.8.1 Modal Analysis

Modal analysis, including proper orthogonal decomposition (POD) and spectral proper orthogonal decomposition (SPOD), provides a robust framework for analyzing turbulence and flow loss mechanisms in turbomachinery components (Schmidt et al. (2020), Lengani et al. (2017) and Lengani et al. (2022)). These methods decompose complex, unsteady flow fields into dominant coherent structures, facilitating the identification of energy-dominant

flow features such as separation zones, wake structures, and turbulence-induced losses. While POD extracts spatial modes with the highest kinetic energy content, SPOD extends this by incorporating frequency-domain analysis to reveal frequency-dependent structures and flow loss mechanisms such as blade passing effects, vortex shedding, and acoustic interactions. The following sections provide a brief overview of these decomposition methods. A more comprehensive description is presented in Appendix A.

2.8.1.1 POD

POD is a statistical tool originally introduced by Lumley (1979) to extract the most energetic modes from turbulent flow fields. It decomposes a fluctuating vector field, $q'(x, t)$, into a series of deterministic spatial functions, $\phi_k(x)$, modulated by time-dependent coefficients, $a_k(t)$. This decomposition can be expressed as

$$q'(x, t) = \sum_{k=1}^{\infty} a_k(t) \phi_k(x), \quad (2.104)$$

where $x = (x, y, z)$ represents the position vector, and $q'(x, t) = q(x, t) - \bar{q}(x)$ is a fluctuating variable (such as a velocity) vector field. POD modes capture distinct fractions of the total turbulent kinetic energy (TKE), with eigenvalues λ_k indicating their respective energy contributions. To efficiently analyze large datasets, the snapshot POD method (Sirovich (1987)) is employed, which reduces computational storage. Usually, a small subset of the most energetic POD modes suffices to reconstruct the dominant flow structures, significantly reducing dataset complexity while preserving essential flow dynamics. The POD procedure is presented in Appendix A.

2.8.1.2 SPOD

SPOD extends POD by incorporating spectral analysis, making it well-suited for examining spatiotemporal coherence in flow data. Unlike standard POD, which considers only spatial orthogonality, SPOD employs Welch's method to estimate the cross-spectral density (CSD) matrix at each frequency, improving spectral estimation accuracy (Towne et al. (2018)). Key steps in SPOD include segmenting the time series into overlapping windows, applying a Fourier transform to each segment, constructing the data matrix $\hat{Q}$, where each column represents a transformed realization, computing the CSD matrix at each frequency, and extracting SPOD modes via eigenvalue decomposition. SPOD combines the optimal energy representation of POD with the frequency-based insights of dynamic mode decomposition (DMD), offering a comprehensive approach for analyzing coherent structures in turbulence. By isolating dominant flow structures, these methods can guide aerodynamic optimizations, improve turbulence modeling, and mitigate performance losses in turbomachinery applications. A more comprehensive description of the SPOD method is presented in Appendix A.

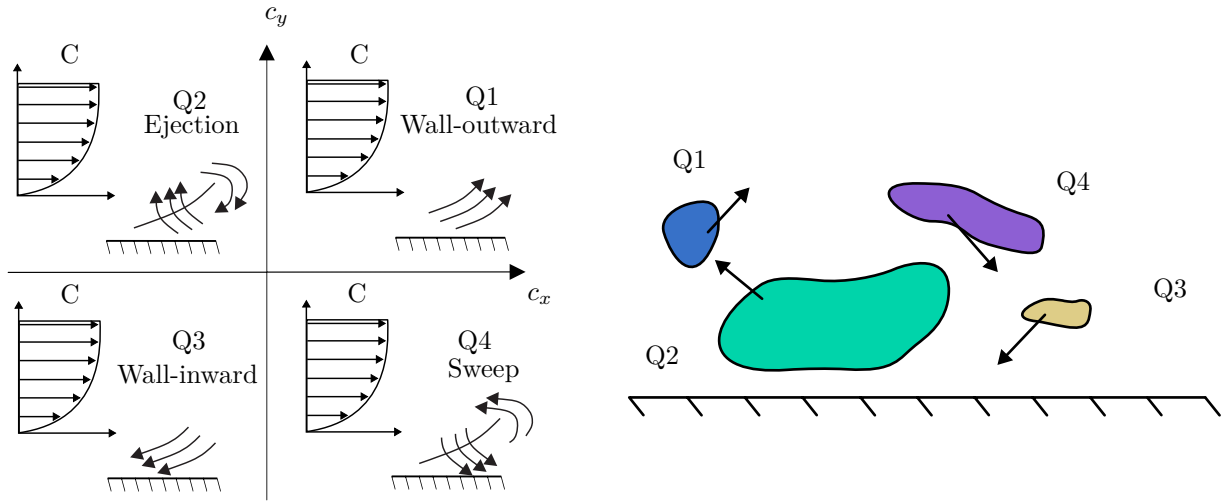


Figure 2.17: Diagram of the categorization of turbulent fluid motion e.g. quadrant events

2.8.2 Quadrant Analysis

Quadrant analysis of the Reynolds stress tensor, first introduced by Wallace et al. (1972), is a valuable technique for identifying the dominant contributions to turbulence. This method enhances the understanding of Reynolds shear stress generation and its role in turbulent kinetic energy production. Since Reynolds shear stresses play a major role in developing turbulent flow, Such insights can improve turbulence modeling and predicting its effects in various engineering applications. In Reynolds stress quadrant analysis, the Reynolds shear stress ($c'_x c'_y$) is categorized into four quadrants based on the sign of the streamwise velocity fluctuation (c'_x) and the wall-normal velocity fluctuation (c'_y).

1. The first quadrant Q1 ($c'_x > 0$, $c'_y > 0$) includes outward motion of high-velocity fluid.
2. The second quadrant, designated Q2 ($c'_x < 0$, $c'_y > 0$), represents the outward motion of low-velocity fluid, commonly associated with turbulent ejection events.
3. The third quadrant Q3 ($c'_x < 0$, $c'_y < 0$), describes the inward motion of low-velocity fluid.
4. The fourth quadrant Q4 ($c'_x > 0$, $c'_y < 0$) represents the inward motion of high-velocity fluid.

Among these, Q2 (ejections) and Q4 (sweeps) are gradient-type motions, meaning they contribute most significantly to the Reynolds shear stress by transporting momentum in the direction of the mean velocity gradient. In contrast, Q1 and Q3 are counter-gradient motions, as they exhibit momentum fluxes that oppose the mean gradient. The term "gradient-type motion" refers to vertical momentum transport caused by fluid elements moving up and down the mean velocity gradient, while "counter-gradient-type motion" implies the opposite effect. Figure 2.17 illustrates the categorization of different quadrant events based on turbulent fluid motions. At any fixed distance from the wall, the fractional contributions of all four quadrants sum to unity, providing a quantitative framework to analyze turbulence dynamics and energy transfer in boundary layers and other shear-driven flows.

3 Literature Review, Motivation and Outline of the Thesis

3.1 Literature review

In this chapter, the focus will be on reviewing research works that have examined the flow physics of turbine and compressor cascades. The objective of this chapter is to provide a comprehensive overview of the literature, with a particular emphasis on works that have investigated the state of the boundary layer and turbulent flow, flow loss mechanisms, and the impact of free-stream turbulence (FST) on the aforementioned parameters in the context of turbine and compressor flow. Furthermore, the literature review will be primarily confined to numerical works, with a special focus on high-fidelity methods.

3.1.1 Role of high-fidelity studies in turbomachinery

Computational fluid dynamics (CFD) simulations have become central to the design process of turbomachinery. Over recent decades, CFD methods have evolved considerably, from the simple throughflow and inviscid calculations of the 1960s to sophisticated three-dimensional unsteady RANS multistage computations by the 2000s. This increase in simulation fidelity has shifted the emphasis from expensive experimental campaigns to detailed numerical studies. For example, the widespread adoption of RANS-based approaches, instead of measurement campaigns, has saved over 30 millions dollars in new compressor designs (according to Fischberg et al. (1996)). Nevertheless, the transition to a CFD-driven design approach was not without challenges. Programs like the PW6000 underscore the risks associated with an over-reliance on early CFD predictions during critical design stages (Przytarski et al. (2020)). As CFD methodologies matured, they became indispensable in the development of aero-engines, forming the foundation of modern design systems when combined with 0D and 1D design tools (Laskowski et al. (2016)). Despite two decades of progress in CFD, the industry has achieved only modest reductions in the need for experimental measurements, primarily due to persistent uncertainties in CFD predictions, especially those related to turbulence modeling. Moreover, the pursuit of marginal efficiency gains (on the order of maximum 1 %) presents another significant challenge for designers (Sandberg et al. (2019)).

Turbulence critically influences turbomachinery flows by affecting transition, separation, mixing, and heat transfer. As Denton (2010) notes, accurate prediction of turbulent phenomena is essential for reliable turbomachinery design. Yet, modeling turbulence remains problematic in areas with strong pressure gradients and pronounced surface curvature. The inherent unsteadiness in turbomachinery flows, arising from both deterministic sources

(such as rotor-stator interactions) and stochastic sources (turbulence itself), further complicates turbulence modeling.

As shown in Fig. 2.3, (U)RANS methods can not resolve or directly model turbulent fluctuations, since their turbulence models are typically validated against canonical cases and may struggle with novel configurations (Tucker (2013)). A critical limitation of URANS methods is the assumption of a spectral gap, which posits that deterministic and stochastic unsteadiness exist on separate scales without significant interaction. This assumption, as discussed by Sandberg et al. (2019) and Tucker (2014), challenges conventional turbulence models even in deterministic flow scenarios.

New design trends that favor higher loading, compactness, and reduced weight, challenge the spectral gap assumptions, potentially introducing errors in stage or multi-stage turbomachinery simulations. According to Sandberg et al. (2019), accurately predicting turbulence and its interaction with deterministic unsteadiness remains the primary challenge for simulation accuracy. High-fidelity methods, such as LES or DNS, can address these challenges by resolving stochastic turbulent behavior, however, they incur a significant computational cost. Estimates by Laskowski et al. (2016) and Gourdain et al. (2014) suggest that the computational cost of DNS/LES can be 100 to 10,000 times higher than that of RANS, depending on the geometry complexity and operating conditions. As a result, high-fidelity simulations are not yet routinely affordable for industrial compressors and turbines. To reduce these costs, hybrid methods, such as hybrid RANS-LES and wall-modeled LES, and more efficient discretization techniques have been developed. Nonetheless, these methods face limitations, including difficulties in predicting transition and numerical instabilities. Despite efforts by researchers such as Menter et al. (2020), it remains unlikely that a universal turbulence model will emerge soon. Instead, high-fidelity simulations provide critical physical insights that can guide the development of physics-driven, lower-order models. The ability to conduct DNS/LES virtual experiments and extract detailed data from three-dimensional flow fields is a crucial step toward this goal.

In academia, high-fidelity simulations of mid-span blade sections and isolated blades are now routinely performed, significantly advancing our understanding of key phenomena such as transition, separation, and mixing in turbomachinery. Although progress is being made toward simulating multi-blade and multi-row effects, simulations of these close-to-real configurations are still limited. Determining the required resolution for accurately capturing physical behavior remains a central challenge, as addressed in several studies (e.g. Choi et al. (2001), Zang et al. (1989), Souza et al. (2005), Celik et al. (2006) Geurts et al. (2002), Pichler et al. (2016) and Rezaeiravesh et al. (2018))

3.1.2 Turbine Cascade Flow Physics

The computational cost of high-fidelity simulations correlates with the Reynolds number (cf. (2.55)), which initially directed research on aero-engine components towards low-pressure turbine (LPT) flows. LPTs generally operate at lower Reynolds numbers, typically between 40,000 and 400,000 (Hodson et al. (2005a)), than other aero-engine components, owing to their smaller blade chord lengths and lower air density. These conditions make LPT flows particularly prone to separation, highlighting the need for precise

boundary layer predictions to minimize profile flow losses and optimize turbine designs. Early high-fidelity investigations focused on unraveling the mechanisms of boundary layer transition on LPT blades, as well as understanding the impact of both stochastic and deterministic unsteadiness on the boundary layer (Wissink et al. (2006), Wu et al. (2001)). Numerous experimental and numerical studies, including LES and DNS, have greatly enhanced our knowledge of transition processes in laminar separation bubbles. Key research areas include transition mechanisms at low free-stream turbulence levels (McAuliffe et al. (2010), Wissink et al. (2004)), bypass transition at elevated free-stream turbulence levels (Đurović et al. (2021)), influence of upstream wakes (bars or stages) on separation bubbles (de Vincentiis et al. (2023), Wu et al. (1999), Wissink (2003)) and role of passive (e.g. roughness elements) and active control (e.g. vortex generator) on separation (Sondergaard et al. (2002)).

On the experimental side, extensive investigations have provided a comprehensive understanding of transition mechanisms in turbomachinery. Notably, research by Halstead et al. (1997), Graveline et al. (2005), Hodson et al. (2005b), Volino et al. (2001) and others has largely relied on linear cascade test rigs, which effectively reproduce characteristics such as wake kinematics and secondary flow systems. Halstead et al. (1997) remains a seminal work for understanding wake-boundary layer interactions, emphasizing how Reynolds number, surface roughness, and FST influence boundary layer development in both turbines and compressors. Many studies have employed linear cascades or flat plates to facilitate controlled measurements (using techniques like particle image velocimetry or hot-wire anemometry) (Volino et al. (2001) and McAuliffe et al. (2010)). Exposing a flat plate to a streamwise pressure gradient helps simulate realistic turbine flows. However, while such configurations yield valuable insights into the fundamentals of transition, they do not fully capture the complexities of real turbomachinery, such as the impact of incoming wakes or flow deflection within blade passages. Consequently, there is a growing focus on high-fidelity numerical studies of realistic LPT configurations, enabled by advancements in computational power. Over the years, several turbulence modeling communities have addressed the challenge of transition prediction. For instance, Launder et al. (2002), Bradshaw (1996) and Spalart et al. (2000) offer a comprehensive review and critical evaluation of various RANS turbulence models, proposing adjustments to enhance their accuracy in both attached and separated flows.

In summary, the literature reveals that accurate prediction of boundary layer transition, especially in laminar separation bubbles (LSBs) observed in LPT flows, is vital for improving turbomachinery performance. The transition process, which may involve turbulent reattachment, is highly dependent on operating conditions and is further influenced by interactions with adjacent blade rows in actual machines. This complex interplay of factors highlights the ongoing need for advanced high-fidelity studies to address remaining research gaps.

Review of transition in attached boundary layers

The transition from laminar to turbulent flow is significantly influenced by free-stream turbulence (FST). Numerous studies (Alam et al. (2000), McAuliffe et al. (2008) and Balzer et al. (2016)) have examined the mechanisms, introduced in Sec. 2.6.1, using flat plates or simplified geometries subjected to streamwise pressure gradients. These investigations have confirmed the phenomenology of bypass transition while highlighting challenges posed

by FST characteristics, such as the integral length scale, where larger scales promote an earlier transition.

Review of transition in separated boundary layers

As already stated in Sec. 2.6.2, there are several mechanism (inviscid Kelvin-Helmholtz instability and Tollmien-Schlichting instability) responsible for the transition in the separated boundary layers. A comprehensive understanding of the interplay between KH and TS instability mechanism and investigating the impact of FST on separation bubble behavior is essential for accurately predicting the behavior of separated shear layers and the associated performance losses in turbomachinery components. Experimental investigations by Saric et al. (2002), Diwan et al. (2009), Simoni et al. (2012), Volino (2002a) and Volino (2002b) have clarified the aforementioned effects, while DNS studies by Zhao et al. (2020b), Kalitzin et al. (2003), Wissink et al. (2004) and Balzer (2011) have provided valuable numerical insights regarding understanding the transition mechanism in separated flows.

Review of influence of inflow turbulence on transition and boundary layer

Raverdy et al. (2003) demonstrated that, in the absence of periodic wakes, the suction-side transition process follows a similar path to that reported by Alam et al. (2000) and discussed earlier. In this scenario, a laminar separation bubble initially forms, which is then destabilized by the Kelvin-Helmholtz instability, eventually leading to turbulent reattachment. Researchers such as Zhao et al. (2020b) and Hosseini et al. (2015) have investigated how both low and high FST levels affect the transition mechanisms in LPTs. Their findings emphasize the persistent challenges posed by separation bubbles, such as increased profile losses or modeling and prediction difficulties, even under highly turbulent conditions. Numerous studies have examined various aspects of LPT flows, focusing on the role of free-stream turbulence and upstream wakes in determining the state of the boundary layer and the associated flow losses. Notable contributions include those by Wu et al. (2001) Michelassi et al. (2002), Wissink (2003), Pichler et al. (2015), Matsuura et al. (2006), Kalitzin et al. (2003), Sarkar (2007), Sarkar (2008), Sarkar (2009), Vadlamani (2014) and Müller-Schindewolffs (2022).

Numerical studies of low-pressure turbine flows often employ quasi-3D (q3D) and less often fully three-dimensional modeling strategies. Research by Wu et al. (2001), Michelassi et al. (2002), Wissink (2003), and Müller-Schindewolffs (2022) has shed light on the roles of FST and upstream wakes in LPT blade flows. Three-dimensional effects, particularly side-wall flows, are crucial in LPT studies because blade, side-wall interactions account for up to 30 %–40 % of total flow losses. High-fidelity investigations by Koschichow et al. (2014), Cui et al. (2016), Cui et al. (2017), Pichler et al. (2018), Winhart (2021), Robison et al. (2021), Donovan (2023) have examined 3D LPT flows in detail. For instance, Koschichow et al. (2014) analyzed the influence of periodic wakes on the side-wall flow and the resulting vortex system. Koschichow et al. (2014) showed, when the incoming boundary layer is thin, the time-averaged effect of the wakes on the side-wall boundary layer and overall secondary flow is also small. Cui et al. (2016), Cui et al. (2017) studied flow loss generation and demonstrated that incoming wake or purge flows can interact with the side-wall boundary layer, increasing flow losses by up to 30 %. Pichler et al. (2018) further highlighted the significant effect of the prescribed side-wall boundary layer on vortex system development and secondary flows. Advances in computational power have enabled more detailed 3D

investigations of side-wall flows, substantially enhancing our understanding of secondary flow systems and vortex dynamics near the side-wall.

The specific test case examined in this work, MTU-T161, has been studied using high-fidelity simulations over recent years by researchers such as Müller-Schindewolf (2022), Morsbach et al. (2023), Fard Afshar et al. (2023a) and Rosenzweig et al. (2023). These investigations have focused on analyzing blade pressure distribution and flow loss mechanisms, with simulation results validated against experimental measurements to confirm the feasibility of the employed numerical methods and codes. These studies also highlight the substantial computational challenges associated with modeling three-dimensional LPT configurations that include side-walls and turbulent inflows. Although many research groups have contributed valuable insights over the past years (as shown in the present chapter), the accurate prediction of the transition process and the influence of free-stream turbulence continue to represent demanding areas of study. Continued advancements in high-fidelity simulation techniques, coupled with their application to realistic LPT configurations, promise to further refine our understanding of transition mechanisms, separation bubble dynamics, and their impact on turbomachinery performance.

3.1.3 Compressor Cascade Flow Physics

Review of transition mechanisms and the influence of free-stream turbulence on compressor flow

Experimental work by Hilgenfeld et al. (2004) and high-fidelity studies by Zaki et al. (2010) on compressor cascades (for example, using the NACA65 blade profile) have been foundational in examining transition mechanisms in compressor flows. These studies demonstrate that transition in compressor flow is highly sensitive to FST levels. In a series of investigations, Zaki et al. (2006), Zaki et al. (2009) and Zaki et al. (2010) explored the influence of FST on transition in NACA65 cascade flows under varying turbulence intensities (from 0% to 10%), revealing that transition mechanisms on both the pressure and suction sides vary with the turbulence level. Under low turbulence conditions, laminar inflow leads to flow separation on both surfaces. At moderate turbulence levels (around 3%), the pressure side boundary layer remains attached, exhibiting characteristics of both natural and bypass transition, while the suction side initially behaves similarly to the laminar case. However, once separation occurs, Kelvin–Helmholtz instabilities trigger turbulence breakdown and subsequent reattachment. In this scenario, prominent streaky structures modulate the instantaneous flow and affect the separation bubble size. Zaki et al. (2010) shows, that at high turbulence levels (6%), both pressure and suction sides exhibit a bypass transition scenario, with stronger streaks on the suction side leading to turbulent spots that locally suppress the separation bubble. At sufficiently high turbulence intensities, the increased frequency of turbulent spots can almost entirely suppress separation.

These findings highlight the complex nature of transition phenomena in compressor flows and underscore the need for turbulence models that account for the sensitivity of transition to varying inflow conditions. Recent DNS studies by Wheeler et al. (2018) on a controlled diffusion aerofoil (CDA) compressor cascade across a range of Reynolds numbers have further emphasized the significance of non-equilibrium behavior in the boundary layer. Non-equilibrium conditions, characterized by a lag between turbulence production

and dissipation, showed to affect flow loss mechanisms by reducing the dissipation coefficient. The work of Wheeler et al. (2018) provides new insights into flow loss prediction in compressor flows and stresses the importance of incorporating non-equilibrium dynamics for more accurate modeling and optimization of compressor designs. While most conventional turbulence models assume a balance between production and dissipation, Wheeler et al. (2018) showed that up to 30 % of the suction-side boundary layer exhibited non-equilibrium characteristics. Similar investigations by Monier et al. (2020), Spencer et al. (2023) and Przytarski et al. (2024) have further focused on improving conventional RANS models or elucidating the origins of these modeling discrepancies.

Review of side-wall Flows and Corner Separation

Predicting the behavior of side-wall flows in compressors is particularly challenging due to two main factors. The thickness of the boundary layer and its tendency to separate under adverse pressure gradients. When side-wall flows separate, they generate complex three-dimensional structures where suction-side separation interacts with the midspan flow. Studies by Lei et al. (2008) and Gao et al. (2015) revealed that compressor corner separations can manifest either as closed or open topologies, or may periodically transition between these states. In a further contribution to understanding compressor corner separation, Scillitoe (2017) examined its dependence on free-stream turbulence levels and the state of the side-wall boundary layer using linear cascades. The results showed that, although the sensitivity of corner separation to free-stream turbulence is somewhat limited, it strongly depends on whether the side-wall boundary layer is laminar or turbulent. In particular, a laminar side-wall boundary layer tends to separate earlier than a turbulent one, leading to the formation of a larger passage vortex and, consequently, higher flow loss rates. These findings underscore the critical need for precise characterization of side-wall boundary layers and high-fidelity modeling to improve compressor performance predictions.

The transitional behavior of compressor boundary layer flow persists even at high Reynolds numbers and elevated free-stream turbulence intensities. Experimental work (Schreiber et al. (2004)) has shown that, even at a relatively high Reynolds number ($Re_c = 8.4 \times 10^5$) and a free-stream turbulence intensity of 2.5%, the boundary layer on the suction surface can remain laminar up to the peak suction over a broad range of incidence angles. It is well documented in the literature (Mayle (1991), Denton (1993)) that laminar separation leads to significant profile flow losses below a critical Reynolds number of 2×10^5 , emphasizing the increased importance of transition effects in the design of smaller engines. Overall, the challenges in studying compressor flows arise from several factors, including the transitional behavior of the boundary layer, high Reynolds numbers, and three-dimensional separation phenomena. A comprehensive understanding of transition mechanisms, turbulence effects, and side-wall boundary layer dynamics is crucial for enhancing compressor performance predictions. Advanced modeling approaches that integrate these phenomena promise to yield more accurate and efficient compressor designs.

3.1.4 Comparison of high-fidelity methods with low order models

The evaluation of RANS and URANS methods against high-fidelity results has become essential for enhancing the predictive capabilities of low-order models in turbomachinery design. With increasing computational power, the role of high-fidelity simulations, has

grown in refining RANS-based models and providing deeper insights into turbulence-driven losses. This comparison has been carried out in the contexts of both low-pressure turbines (LPT) and high-pressure compressors (HPC), which are the focus of the current work.

Low-Pressure Turbine

1. Wu et al. (2001) used the outcomes of wake-induced boundary layer transition on flat plates to assess several turbulence models, finding that models like the $\nu^2 - f$ model successfully captured the flow features associated with wake-induced transition.
2. Michelassi et al. (2002) extended the analysis (i.e. comparison between RANS and LES) to turbine cascades and demonstrated significant challenges in accurately capturing wake dynamics on the blade suction side due to incorrect turbulence production and diffusion in RANS.
3. Sandberg et al. (2015) demonstrated that DNS captures transitional and turbulent structures more accurately than RANS, highlighting that RANS struggles in flows dominated by free-stream turbulence and separated shear layers.

Limitations of linear eddy viscosity models

Several studies have examined the validity of the Boussinesq (Boussinesq (1877)) eddy viscosity assumption. Linear eddy viscosity turbulence models inherently struggle in regions with strong turbulence anisotropy, such as at the leading and trailing edges and within boundary layers, where the stress-strain relationship is non-linear and requires advanced turbulence closures.

1. Michelassi et al. (2002) demonstrated that near separation regions, RANS models underestimate turbulent kinetic energy (TKE) due to their inability to resolve the high-frequency unsteady structures present in wake interactions.
2. Pichler et al. (2018) further demonstrated that while RANS significantly underpredicts wake mixing, resulting in a slower decay of wakes, it still performs adequately for capturing overall flow structures, secondary flows, and flow loss distributions on individual turbine blades.

These findings emphasize the need for high-fidelity data to refine lower-order models and enhance their reliability in LPT applications.

High-Pressure Compressor

The high computational cost of well-resolved high-fidelity simulations for compressors remains a significant barrier to their widespread industrial use. This limitation has led to an increased number of under-resolved high-fidelity simulations that, although promising, still offer only limited advantages over current URANS methods for design purposes. For instance, Gourdain et al. (2014) conducted one of the most extensive compressor LES studies (wall modeled LES), simulating a full compressor stage at $Re \approx 700k$ with nearly 860 million grid cells, yet still not meeting the ideal high-fidelity grid requirements. Comparisons with experimental data revealed that LES provides only modest improvements over URANS in predicting overall pressure ratio and efficiency. However, LES excels in capturing wake evolution and boundary layer behavior, as observed in LPT studies. Gourdain et al. (2014) highlights a fundamental limitation of URANS methods in resolving the complex interplay between deterministic and stochastic interactions, particularly the accurate

modeling of boundary layer transition. Such inaccuracies account for efficiency differences of more than 2 % between URANS and LES, primarily because URANS typically assumes a fully turbulent boundary layer. Leggett et al. (2021) compared LES and RANS for a NACA65 cascade at $Re \approx 300k$ across various incidence angles. While RANS was able to predict overall flow losses and pressure distributions reasonably well, it struggled to accurately capture wake shapes and their evolution. These shortcomings are especially critical in multi-stage compressor environments, where wake interactions significantly influence overall performance. Collectively, these studies suggest that while (U)RANS methods can provide acceptable predictions for global performance metrics, they fall short in resolving detailed flow physics, such as wake interactions. This limitation underscores the superior insights provided by LES, which are critical for advancing compressor design.

Machine Learning in Turbulence Modeling

In recent years, machine learning (ML) has emerged as a powerful tool for enhancing turbulence modeling in RANS simulations by leveraging high-fidelity datasets from LES and DNS for training. The primary objective of integrating ML into turbulence modeling is to overcome the long-standing limitations of RANS models, particularly their challenges in accurately capturing anisotropy, non-equilibrium effects, and flow separation under complex flow conditions, all while maintaining computational efficiency. By incorporating ML techniques, researchers aim to significantly improve the predictive accuracy of low-order models, ultimately leading to more reliable and efficient turbomachinery designs. This rapidly growing field has attracted considerable attention from various research groups. Studies such as Hammond et al. (2022), Zou et al. (2024), Zhu et al. (2019) and Duraisamy (2021) provide comprehensive overviews of recent advancements and key developments in ML-driven turbulence modeling.

3.1.5 Flow loss analysis using high-fidelity methods

Accurately predicting flow losses is essential for utilizing high-fidelity methods as reliable design tools in turbomachinery. Over the past decade, these advanced methods have been extensively used to evaluate performance, investigate flow loss mechanisms, and refine flow loss quantification metrics. By providing a detailed representation of the entire flow, high-fidelity simulations allow for a deeper analysis of how flow losses are generated and distributed. Chen et al. (2015) examined how inflow turbulence, in the form of wakes with varying frequencies, influences blade flow losses in LPTs using DNS. Michelassi et al. (2015) expanded upon this research by studying attached and separated flow conditions, identifying the dominant mechanisms responsible for flow losses in each case. Total pressure loss remains the most widely used metric in turbomachinery studies, largely because it can be directly measured in experimental investigations. However, despite its usefulness, the total pressure loss metric does not fully capture certain effects, such as thermodynamic interactions or inefficiencies in energy conversion, which are crucial for a complete assessment of performance.

To overcome the limitations of conventional metrics, alternative approaches such as mechanical work potential (Miller (2013)) and entropy generation (Zhao et al. (2020b)) have gained prominence in recent research. Although these methods are more challenging to

measure directly, they provide a more comprehensive understanding of flow loss mechanisms. Miller (2013) introduced a framework for quantifying energy losses in turbomachinery using mechanical work potential, a metric that represents the maximum extractable useful work from a flow. This approach enables engineers to attribute energy losses to specific factors such as turbulence or pressure drops while also incorporating thermodynamic effects, offering a broader perspective on flow efficiency. Building on this concept, Leggett et al. (2020) demonstrated the effectiveness of mechanical work potential analysis in compressor cascades using LES, showing its value in evaluating performance, particularly in unsteady flow conditions.

Another significant approach to flow loss quantification is entropy generation. Denton (1993) provided a fundamental analysis of the physical origins and consequences of flow losses in turbomachinery, emphasizing the role of entropy generation as a key indicator of flow irreversibilities. His work laid the foundation for further research into optimizing designs and reducing inefficiencies. More recent studies have refined this concept through the development of entropy generation rate transport equations, which establish a direct link between entropy generation, viscous dissipation, and heat transfer. For instance, Zhao et al. (2020b), Rosenzweig et al. (2023) and Wheeler et al. (2016) used LES and DNS to analyze entropy generation in low-pressure and high-pressure turbines, while Leggett et al. (2021), Przytarski et al. (2020), Przytarski et al. (2021), Przytarski et al. (2024) and Borchert et al. (2023) extended similar analyses to compressor cascades, reinforcing the importance of entropy-based metrics for assessing and mitigating flow losses.

High-fidelity simulations offer deeper insights into flow loss generation mechanisms, surpassing traditional metrics like total pressure loss in capturing flow complexities. By incorporating advanced metrics such as mechanical work potential and entropy generation, researchers can achieve a more accurate representation of flow loss sources, leading to improved turbomachinery performance and optimization strategies. The integration of these sophisticated analytical techniques with high-fidelity modeling is essential for addressing complex aerodynamic and thermodynamic interactions, ultimately enhancing predictive accuracy and design efficiency.

3.2 Motivation

In recent decades, research on turbine and compressor cascades has primarily focused on understanding the fundamental principles of fluid flow. Numerous studies have explored the influence of inflow turbulence on transition processes and flow loss mechanisms. As outlined in Sec. 3.1, the use of high-fidelity numerical methods in low-pressure turbine cascade studies has grown rapidly, demonstrating their potential to significantly enhance design methodologies. However, despite these advancements, their integration into industrial applications remains limited. One major obstacle is the lack of validation under realistic conditions, which undermines confidence in their predictive accuracy and reliability. High-fidelity simulations of realistic turbomachinery configurations are still relatively limited in the literature. To develop a more representative computational model, it is essential to account for side-wall effects and free-stream turbulence (FST), as these factors play a critical role in transition processes and flow loss mechanisms. Moreover, the validation of high-fidelity simulations at off-design operating conditions, where traditional

RANS turbulence models lose accuracy, has not been adequately addressed. Additionally, while recent advancements introduce more sophisticated flow loss analysis techniques, these methods have yet to be systematically applied to realistic turbomachinery flows. Although turbulent flows have been extensively studied in canonical test cases, transitional flows, which are prevalent in turbomachinery applications, remain comparatively underexplored. In particular, the application of statistical methods to transitional flows has not been systematically examined. This thesis aims to bridge these research gaps by conducting high-fidelity simulations of realistic turbomachinery flows, specifically, a highly loaded low-pressure turbine cascade and a high-pressure compressor cascade, while providing comprehensive validation. Additionally, the influence of free-stream turbulence on the transition process and flow loss mechanisms will be systematically analyzed for these test cases, offering new insights into the complex aerodynamics of turbomachinery components.

3.3 Outline of the thesis

Building on the research gaps identified earlier, this thesis investigates turbulence behavior, flow loss mechanisms, and transition in a highly loaded low-pressure turbine (LPT) cascade and a high-pressure compressor (HPC) cascade using high-fidelity simulations (LES/implicit LES). These test cases are explored under realistic, off-design conditions, with a particular focus on free-stream turbulence (FST) effects and modeling discrepancies in industrial CFD workflows. The key objectives are listed below.

1. Validation of large eddy simulation (LES) for mid-span flow in LPT and HPC cascades

The accuracy of LES in predicting mid-span aerodynamic behavior is evaluated by comparing simulation results with experimental data. Key design parameters such as blade pressure distribution and wake flow loss coefficients are validated. A systematic mesh sensitivity study is performed to quantify the influence of grid resolution on the results. Furthermore, different numerical approaches, including finite volume (FV) and discontinuous Galerkin spectral element methods (DGSEM), are assessed to determine their applicability for industrial high-fidelity CFD applications.

2. Quantification of the discrepancies between high-fidelity LES and conventional RANS turbulence models

The predictive capabilities of industry-standard RANS models are compared against LES results at off-design conditions. The objective is to quantify systematic deviations in boundary layer development and turbulence levels, wake flow loss predictions and identify the root causes of RANS underperformance in LPT flows (transition and separation) and HPC flows (adverse pressure gradients and turbulence anisotropy).

3. Identification of flow loss mechanisms using advanced entropy-based analysis

Flow loss mechanisms in LPT and HPC cascades are investigated using entropy-based methods. The study aims to determine the primary sources of aerodynamic flow losses by analyzing turbulence production and entropy generation. A particular focus is placed on the role of separated shear layers in LPT flows and turbulent boundary layers in HPC flows. The objective is to establish a direct correlation between

turbulence structures and flow loss generation, providing a refined understanding of efficiency losses in turbomachinery components.

4. Determination of the influence of free-stream turbulence (FST) on transition and flow loss mechanisms

A systematic study is conducted to quantify the effects of different levels of free-stream turbulence intensity on boundary layer development, separation behavior, and wake flow losses. The objective is to determine how increasing FST affects transition onset, separation bubble size, and overall flow loss generation, particularly in off-design operating conditions. The findings aim to provide practical insights into the design of turbomachinery components that are more resilient to inflow turbulence variations.

This thesis addresses the above objectives, thereby contributing to the bridge between academic research and industrial CFD applications. It lays the foundation for improving the accuracy of turbulence modeling and enables the development of more reliable predictive tools for next-generation aero-engines and gas turbines.

4 Numerical Methods

In this chapter, the various spatial discretization methods employed in this study are introduced. In addition, the boundary conditions and temporal discretizations are briefly discussed. Moreover, the error analysis methods employed in this study are reviewed. In conclusion, the test cases examined in the present work are presented. For each test case, a validation is conducted using a mesh study.

4.1 Spatial discretization

The solution of the previously introduced Navier-Stokes equation system represents the flow. However, since this system consists solely of differential equations, solving it analytically is generally infeasible due to its complexity. As a result, various numerical methods have been developed to approximate the solution, as previously discussed. Among these, the finite volume method (FVM) is currently the most widely used approach in fluid mechanics. The process of converting the general system of conservation equations into a global system of equations, based on a finite number of computational cells, is known as discretization. According to Hirsch (2007), discretization can be divided into two main steps as follows

1. Discretization of the flow domain, also called mesh generation.
2. Discretization of the equations.

Mesh generation plays a vital role in ensuring the accuracy of high-fidelity simulations. To capture the physics reliably, the spatial resolution, and thus the number of computational cells, must be sufficiently fine to directly resolve large-scale, highly anisotropic eddy structures, while ensuring that smaller, unresolved turbulent scales fall within the universal equilibrium range of the turbulence model. Since the nature of large-scale vortex structures varies with each specific case, a mesh sensitivity study was performed for both test cases presented in this work. The specific approach for spatial discretization depends on the numerical method employed. In this work, either the finite volume (FV) method or the discontinuous Galerkin (DG) method is used, both of which have been introduced in the corresponding sections. For the simulations presented in this work, the geometries were meshed using PyMesh, a meshing tool developed by the German Aerospace Center (DLR) (Weber et al. (2016)) and Autogrid5TM16.1. Both meshing programs are block-structured meshing tools.

4.1.1 Finite Volume discretization

The TRACE finite volume implementation employs a cell-centered finite volume method (LeVeque (2012)) to solve the Favre-averaged Navier-Stokes equations. In this approach, the computational grid is defined by cell vertices, while the flow variables are stored at the cell centers. Ghost cells are introduced to account for the flow states of neighboring blocks and to enforce boundary conditions at interfaces or solid walls. The governing equation has to be fulfilled for each control volume V with surface ∂V is expressed as follows

$$\int_V \frac{\partial \mathbf{q}}{\partial t} dV + \oint_{\partial V} (\mathbf{F} \cdot \mathbf{n}) dA = \oint_{\partial V} (\mathbf{F}_v \cdot \mathbf{n}) dA + \int_V \mathbf{S} dV, \quad (4.1)$$

where q is the vector of conservative solution variables

$$\mathbf{q} = \left(\bar{\rho}, \bar{\rho}\tilde{c}_1, \bar{\rho}\tilde{c}_2, \bar{\rho}\tilde{c}_3, \bar{\rho}\tilde{E}, \bar{\rho}\phi_1, \dots, \bar{\rho}\phi_N \right)^T. \quad (4.2)$$

In (4.1), the left-hand side represents the local temporal changes of the conservative variables and the inviscid flux $\mathbf{F}$. The right-hand side contains the viscous flux vectors $\mathbf{F}_v$ as well as the volume source term $\mathbf{S}$. In a finite volume scheme, the discretized form of the integral equation (4.1) is solved for each control volume. The volume integrals are approximated assuming a constant value of the integrand over the control volume, while the surface integrals are evaluated as a sum over all N_F cell faces with a constant value assigned to each face as in

$$\oint_{\partial V} (\mathbf{F} \cdot \mathbf{n}) dA = \sum_{f=1}^{N_F} (\mathbf{F} \cdot \mathbf{n}_f) A_f. \quad (4.3)$$

In this representation, the cell face f is represented by two vectors. The face normal vector ($\mathbf{n}_f$) and the face area vector (A_f). In order to discretize the inviscid convection terms, a flux difference splitting method is employed, utilizing the second-order accurate Roe (1981) scheme. The gradients of velocity and temperature utilized in viscous terms are calculated using a second-order accurate central differences flux difference splitting method. The system of conservation equations is spatially discretized in terms of the values of the respective conserved variables on the cell surfaces, denoted by q in (4.2).

4.1.2 High order meshes

The discontinuous Galerkin method requires an accurate high-order approximation of geometric boundaries. In this work, the high-order approximation of Cartesian coordinates is derived from a refined linear mesh generated using the block-structured meshing tool PyMesh. The spectral element approach employed in this work necessitates a numerical grid consisting of hexahedral elements. These elements facilitate the tensor-product space formulation, leveraging Lagrange polynomials for interpolation (Bassi et al. (1997b)). The order of the elements influences the number of discarded inner grid points, which in turn affects the vertex coordinates used to define the geometric mapping function. This mapping function is constructed using Lagrange polynomials with equidistant nodes. The

metric terms at solution nodes are subsequently computed within the solver, following the methodology outlined in Sec. 2.5.2.2. A key advantage of this approach is that the geometric approximation order, denoted as Q , is independent of the solution approximation order, allowing user-defined flexibility. Furthermore, the grid stretching typically applied in boundary layers is also utilized for node distribution within high-order cells, leading to enhanced accuracy. Using a piecewise linear representation of solid walls is insufficient for higher-order schemes. Studies have demonstrated that employing straight edges can degrade solution accuracy in the vicinity of walls, even when highly refined grids are used (Bassi et al. (1997a)). To mitigate this issue, a higher-order representation of geometry is essential.

4.1.3 Spatial discretization requirements

To ensure numerical accuracy and capture relevant flow physics, a grid sensitivity analysis is conducted, taking into account established resolution criteria from literature, including Tyacke et al. (2015) and Tucker (2014). The resolution requirements for various numerical methods are summarized in Tab. 4.1. Where Δx^+ , $\Delta y_{\min}^+$ and Δz^+ correspond to the respective wall units in the tangent, normal and spanwise directions of the blade surface. N is the number of grid points.

Method	$\Delta x^+/\Delta y_{\min}^+/\Delta z^+$	N below $y^+ = 10$
DNS	10 – 15/1/5	3 – 5
Wall resolved LES	50 – 130/1/15 – 40	3 – 5
Wall modeled LES	100 – 600/30 – 150/100 – 300	-
Hybrid RANS-LES	100 – 600/1/100 – 300	2 – 5
RANS	1000/1	2 – 5

Table 4.1: Resolution requirements for various numerical methods according to Tyacke et al. (2015) and Tucker (2014)

However, these resolution guidelines are mostly derived from flat-plate configurations and simple flows. In complex geometries, such as those considered in this work, which involve separating shear layers and massive separation, these criteria become less definitive. Hence, a case-dependent mesh sensitivity analysis is necessary. As stated by Tyacke et al. (2015), for numerical schemes of higher order than 2, these values would represent conservative estimates. For more specific applications, such as low-pressure turbine (LPT) cascades, Sandberg et al. (2019) provide refined resolution criteria, especially for low Reynolds number flows $Re < 100,000$, where the boundary layer is transitional and separation bubbles may occur. The resolution requirements, corresponding to wall-resolved LES or coarse DNS, for such flow scenarios may be more restrictive, as shown below

$$\begin{aligned} 50 \leq \Delta x^+ \leq 30, \quad \Delta y_{\text{wall}}^+ < 1 \\ 15 \leq \Delta z^+ \leq 20 \end{aligned} \tag{4.4}$$

Much of the computational cost in these cases arises from the fine resolution requirements in the near-wall region. Similar to RANS, LES requires resolving steep velocity gradients near walls, necessitating fine wall-normal spacing $y^+ \approx 1$ and at least five grid points within the viscous sublayer. In the near-wall region ($y^+ < 100$), turbulent structures are small and turbulence consists of elongated coherent structures (streaks), which are critical for momentum transfer and turbulent energy production. According to Scillitoe (2017), these streaks typically measure around 2000 wall units in the streamwise direction, with a width of 20–80 wall units and a spacing of about 100 wall units in the spanwise direction. At the same time, coarse LES grids can negatively impact solution accuracy by an under-predicted turbulent production, leading to lower Reynolds stresses and skin-friction. It can also cause an incorrect representation of the logarithmic velocity law and excessive anisotropy near the wall. The grid resolution, LES accuracy and the resulting effects on the solution are summarized in Tab. 4.2.

Grid resolution parameters	Quality of LES	Effect on Solution
$x^+ \leq 50, z^+ \leq 12$	High resolution	Generally good predictions of skin friction
$x^+ \leq 100, z^+ \leq 30$	Medium resolution	Thicker and shorter streaks, error in skin-friction prediction
$x^+ \geq 100, z^+ \geq 30$	Poor resolution	Unrealistic streaks, larger error in skin friction prediction

Table 4.2: Grid resolution and LES accuracy from Sagaut (2001) and Scillitoe (2017)

As Reynolds numbers increase, near-wall streaks become smaller, significantly increasing computational costs for LES. Choi et al. (2012) showed that the number of grid points required scales with $Re^{1.8}$ (or $Re^{1.86}$) in the near-wall region, while the points required for the rest of the boundary layer scale as $Re^{0.4}$. This scaling implies that LES becomes much more expensive at higher Reynolds numbers, as seen in gas turbine engines, where the Reynolds number varies significantly across different components. For high Reynolds number flows (10^6), nearly 99% of grid points in LES are dedicated to resolving the near-wall region, even though it constitutes less than 10% of the boundary layer thickness. To reduce computational costs, hybrid RANS-LES (HRLES) approaches are often employed, using RANS near walls and LES in the outer flow. However, for transitional flows, such as those in this work, special treatment of transition zones is required. The suitability and cost-effectiveness of hybrid RANS-LES methods is discussed in the corresponding publications Fard Afshar et al. (2023b) and Fard Afshar et al. (2024).

4.2 Boundary condition

In TRACE, a one-dimensional characteristic non-reflecting boundary condition is applied at the domain outlet to enforce the prescribed time- and surface-averaged boundary state (usually the static pressure) (Schlüss et al. (2016)). In case of FV-LES simulations, a Riemann boundary condition (Roe (1981)) is applied at the domain inlet to set the total pressure, total temperature and flow angles. In case of DG-LES simulations, the one-dimensional characteristic non-reflecting boundary condition (Schlüss et al. (2016)) is used

at both domain inlet and outlet surfaces. To reproduce realistic operating conditions, synthetic turbulent fluctuations in the inflow are introduced. The inlet fluctuation field in TRACE is represented by the synthetic turbulence generator through the superposition of Fourier modes based on the prescribed turbulent length scale and Reynolds stress tensor components (Shur et al. (2014)).

4.2.1 3D inflow conditions including side-walls

To ensure an accurate representation of realistic operating conditions, it is essential to prescribe appropriate inflow conditions, including the development of side-wall boundary layers, to achieve a good agreement with experimental measurements. In addition, the decay of free-stream turbulence must be consistent with the experimental measurements at the specified axial locations upstream of the blade at mid-span. However, measurement data is often not sufficiently dense to be directly utilized as inflow conditions for simulations. Moreover, depending on the method used for synthetic turbulence generation, a certain flow development length is required for turbulence to fully adjust and stabilize. To address these challenges, a precursor channel flow simulation is typically conducted. This allows for the iterative adjustment of both the incoming free-stream turbulence and the inflow boundary layer to ensure consistency with experimental data.

The three-dimensional boundary conditions at the inlet surface are prescribed using spanwise varying profiles of total pressure $p_{t,1}(z)$, total temperature $T_{t,1}(z)$, Reynolds stress tensor $\overline{c'_i c'_j}(z)$ and turbulent length scale $L_T(z)$, while maintaining a constant inflow angle α_1 . At the mid-span of the domain, the operating point should be correctly reproduced, as described later in this section. The spanwise variation of inflow profiles is derived by combining boundary layer profiles from a DNS database (Schlatter (2012)), for a comparable boundary layer, with mid-span operating conditions. These datasets are then scaled to ensure consistency between the boundary layer properties and the mid-span flow. Within the present simulations, the spanwise distribution of total temperature $T_t(z)$ and total pressure $p_t(z)$ is computed, while assuming certain values throughout the boundary layer. By employing the isentropic flow equation, the Mach number distribution is related to the total temperature and total pressure variations as follows

$$\frac{T_t(z)}{T_1} = \left(\frac{p_t(z)}{p_1} \right)^{\frac{\gamma-1}{\gamma}} = 1 + \frac{\gamma-1}{2} M(z)^2. \quad (4.5)$$

The Mach number at the centerline ($M_{1,\text{center}}$) defines the static flow properties at the side-walls. The Mach number distribution itself is obtained from experimental data and DNS results within the boundary layer.

The next step involves specifying the turbulent quantities across the inflow plane. The integral turbulent length scale within the boundary layer is estimated based on the turbulent kinetic energy and its dissipation rate. The prescribed free-stream turbulence intensity and integral length scale must be selected to ensure that the turbulent decay matches experimental observations. In many cases, the measured integral length scales are comparable to or even exceed the blade pitch of the cascade. This would necessitate simulating multiple blade passages, significantly increasing computational costs. To mitigate this issue, an alternative approach is adopted in this work. The components of the Reynolds

stress tensor at the domain inlet are increased while the integral length scale is reduced. This ensures that the turbulence intensity at the cascade leading edge matches experimental observations. Since a lower integral length scale leads to enhanced turbulent decay, this approach effectively reproduces the desired turbulence characteristics while maintaining computational efficiency. As with total pressure and temperature, the turbulent properties are adjusted to ensure a smooth transition between the free-stream values and the boundary layer region.

Figure 4.1 presents the results of the channel-flow simulations conducted to prescribe appropriate inlet boundary conditions in terms of Reynolds stresses, such that the experimentally observed free-stream turbulence decay is accurately reproduced. The computational domain of the channel is extended by 1.5 times the axial chord length of each cascade, however, it does not reach the location of the first measurement point in order to limit computational cost.

Reynolds stress tensor 1 corresponds to the decay obtained for a prescribed isotropic turbulence state with specific normal Reynolds stress components ($\overline{c'_i c'_i} = \overline{c'_j c'_j} = \overline{c'_k c'_k}$). Reynolds stress tensor 2 represents the resulting decay for increased (relative to Reynolds Stress 1) normal Reynolds stress components, while maintaining isotropy ($\overline{c'_i c'_i} = \overline{c'_j c'_j} = \overline{c'_k c'_k}$). The results of Reynolds stress tensor 2 match the experimental results very well at the second measurement point and the leading edge.

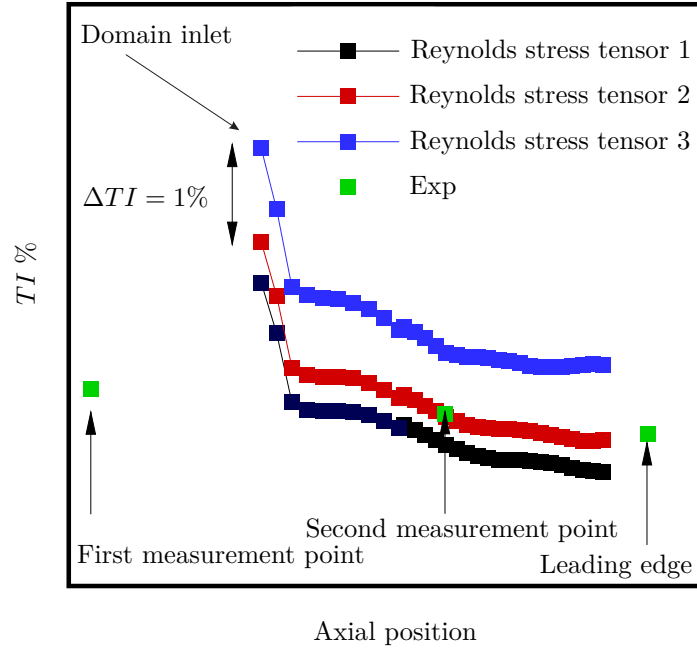


Figure 4.1: Turbulent intensity decay for different boundary conditions e.g. normal Reynolds stresses at the centerline of the channel. The experimental values are shown for two measurement positions before the leading edge. The final measurement point (the green point on the far right) is extrapolated.

In addition to the aforementioned criteria, the accurate development of the side-wall boundary layers is essential to ensure physically consistent flow behavior and meaningful comparison with experimental measurements. For this purpose, the momentum thickness and velocity profile development of the incoming end wall boundary layers should match the experimental values. The above mentioned approach is outlined in more detail for the LPT-cascade in Morsbach et al. (2023).

4.3 Temporal discretization

In addition to spatial differentials and integrals, the governing equations also contain partial derivatives with respect to time, which must be discretized for numerical integration. Similar to spatial discretization, temporal discretization transforms the continuous equations into a form suitable for computation.

In the current work, explicit time integration schemes are employed to solve the semi-discrete formulation of the finite volume and the discontinuous Galerkin spectral element method (DGSEM) (Altmann et al. (2013), Hindenlang et al. (2012)). Due to their locality and simplicity, explicit schemes enhance the scalability of DG methods, which rely on compact stencils for large-scale simulations. As a result, explicit time integration can outperform implicit methods despite their stringent time-step constraints. Furthermore, well-resolved high-fidelity simulations depend not only on the spatial resolution of small-scale structures but also on accurate temporal resolution. Large time-step sizes in implicit schemes can introduce resolution errors, whereas explicit schemes, despite their restrictions on time-step size, inherently maintain the required temporal accuracy. Since the accuracy and stability of high-fidelity simulations for turbomachinery cascade flows are ensured in this work, a three-stage, third-order strong-stability preserving Runge-Kutta scheme described in Shu et al. (1988) is utilized.

The time-step size is constrained by the Courant-Friedrichs-Lewy (CFL) condition, which must generally remain below one to ensure stability in explicit time integration schemes. In TRACE, the time-step size remains constant throughout the simulation. This is particularly advantageous for ensuring consistency with predefined sample intervals of the simulation output (such as three-dimensional statistics and three-dimensional instantaneous fields) and the stabilization period required for non-reflecting boundary conditions. Additionally, once the simulation is properly initialized, the flow conditions remain largely unchanged, leading to a nearly constant time-step restriction.

4.4 Error analysis

Numerical simulations can only provide an approximation of a given fluid mechanics problem. As stated by Ferziger (2008), numerical simulations are inherently subject to uncertainties. Therefore, quantifying numerical errors is a crucial aspect of numerical investigations. In principle, numerical errors can be classified in different ways, depending on the application. Various types of errors play a significant role, particularly in high-fidelity

simulations. The following sections categorize these errors with a focus on their impact on such simulations.

Discretization error

Discretization errors arise from the numerical approximation of continuous conservation equations and are primarily influenced by the computational mesh and spatial discretization methods. In high-fidelity simulations, these errors are a major concern, as accurate resolution of turbulent structures is essential. Insufficient discretization can act as a low-pass filter, damping turbulence, or, in extreme cases, lead to numerical instabilities. The role of discretization has been discussed in detail in Sec. 4.1. In this work, a grid sensitivity analysis is conducted for each simulation, as presented in the corresponding sections.

Boundary condition error

Boundary condition errors encompass all inaccuracies related to the input parameters of the flow solver. These errors include errors stemming from numerical approximations of the exact geometry and the specification of appropriate boundary conditions. In scale-resolving simulations, boundary conditions must ensure that the operating point under investigation is both accurately captured and maintained. Only under these conditions can numerical results be reliably compared with experimental measurements. In this work, flux-averaged mass flow, mass flow difference, pressure ratio, Reynolds number, and Mach number are monitored to assess boundary condition accuracy. An example of monitoring the axial development of two important variables (Reynolds number and Mach number) along a streamline in the middle of the passage is shown in Fig. D.1. It is important to hold these variables during the computation, since they are representative for the corresponding operating point.

Residual error

The system of equations in numerical simulations is solved iteratively, meaning the computed solution never exactly matches the true solution. As the number of iterations increases, the numerical error decreases. The iterative process is typically terminated when the residual, defined as the difference between solutions from consecutive iteration steps, falls below a predefined threshold. The remaining error is the deviation between the exact solution and the iteratively computed result. An example of how boundary condition and residual errors are monitored is provided in Fig. D.2.

Averaging error

In transient, unsteady simulations, particularly in high-fidelity simulations, iteration errors are further compounded by the presence of averaging errors. Similar to flow measurements, individual solutions from iteratively converged time steps are averaged to obtain statistical quantities. The resulting mean value converges toward a final value over time, and the averaging error represents the deviation of this mean from the theoretical final value obtained from infinitely long averaging. More specifically, two types of averaging-related errors must be accounted for and either minimized or estimated

1. Before statistical sampling can begin, the flow must transition from the initial steady state to a fully developed, statistically stationary state. All simulations in this work are initialized using the RANS solution of the setup. If the averaging process starts too early, transient phenomena can significantly distort the final statistical results.

Conversely, if the sampling begins too late, computational resources are unnecessarily wasted.

Therefore, detecting the initial transient phase reliably is essential in every high-fidelity simulation. Figure 4.2 illustrates an example of monitoring the initial transient using the mean squared error (MSE) method introduced by Bergmann et al. (2022) and Morsbach et al. (2023) (cf. Appendix D). The figure presents time series data for axial velocity and pressure at a probe location, along with the truncated MSE computed for multiple probes. The red curves depict the behavior of the truncated MSE for ten probes, primarily located in the wake region. The flow in the wake region has been shown to require the longest convergence duration. The vertical lines mark the minimum values of each curve, which serve as indicators for determining the initial transient phase.

It is important to note that the duration of the initial transient varies depending on the location within the flow and the physical quantity being evaluated. Nevertheless, this method provides a reliable estimation of the required simulation time for the initial transient to pass.

2. The second source of uncertainty concerns the duration of the averaging process for chaotic flow behavior (averaging duration). Due to computational constraints, statistical averaging can only be performed over a finite number of samples, which introduces a degree of statistical error. This error can be particularly significant for higher-order moments or quantities dominated by low-frequency fluctuations, as they require longer averaging periods for accurate estimation.

The impact of both error sources depends on the specific quantity being analyzed and its location within the flow, as these factors influence local timescales. Rather than extending sampling times indefinitely to mitigate these uncertainties, in this thesis, the method introduced by Bergmann et al. (2022) is used to remove initial transients from the time series and estimate statistical errors. An example of how to monitor the convergence is shown in Appendix D. This method has been thoroughly validated for similar cases, as demonstrated in the recent publications (Bergmann et al. (2022)).

Model error

The mathematical description of fluid flows relies on several model assumptions, including Sutherland’s law (Sutherland (1893)), the Boussinesq hypothesis (Boussinesq (1877)), as well as fundamental conservation and state equations. Additionally, various models exist for predicting turbulence and mixing processes. In the aerodynamics of low-pressure turbines and high-pressure compressors, turbulence modeling represents the most significant source of model error (Pichler et al. (2015)). With the exception of Implicit LES and DNS, all numerical simulations introduce turbulence modeling errors. In the extreme case of one- or two-equation RANS models, all turbulence characteristics are reduced to a single scalar value, the turbulent viscosity μ_t . As a result, turbulence modeling errors are particularly dominant in industrial simulations, where obtaining a numerical solution within a short time frame is often prioritized for economic reasons. Due to these efficiency constraints, directly resolving turbulence remains impractical.

As noted by Ferziger (2008), estimating model errors is inherently challenging and requires a detailed comparison of simulation results against a large dataset of high-fidelity experimental measurements. All numerical simulations performed in this work, including

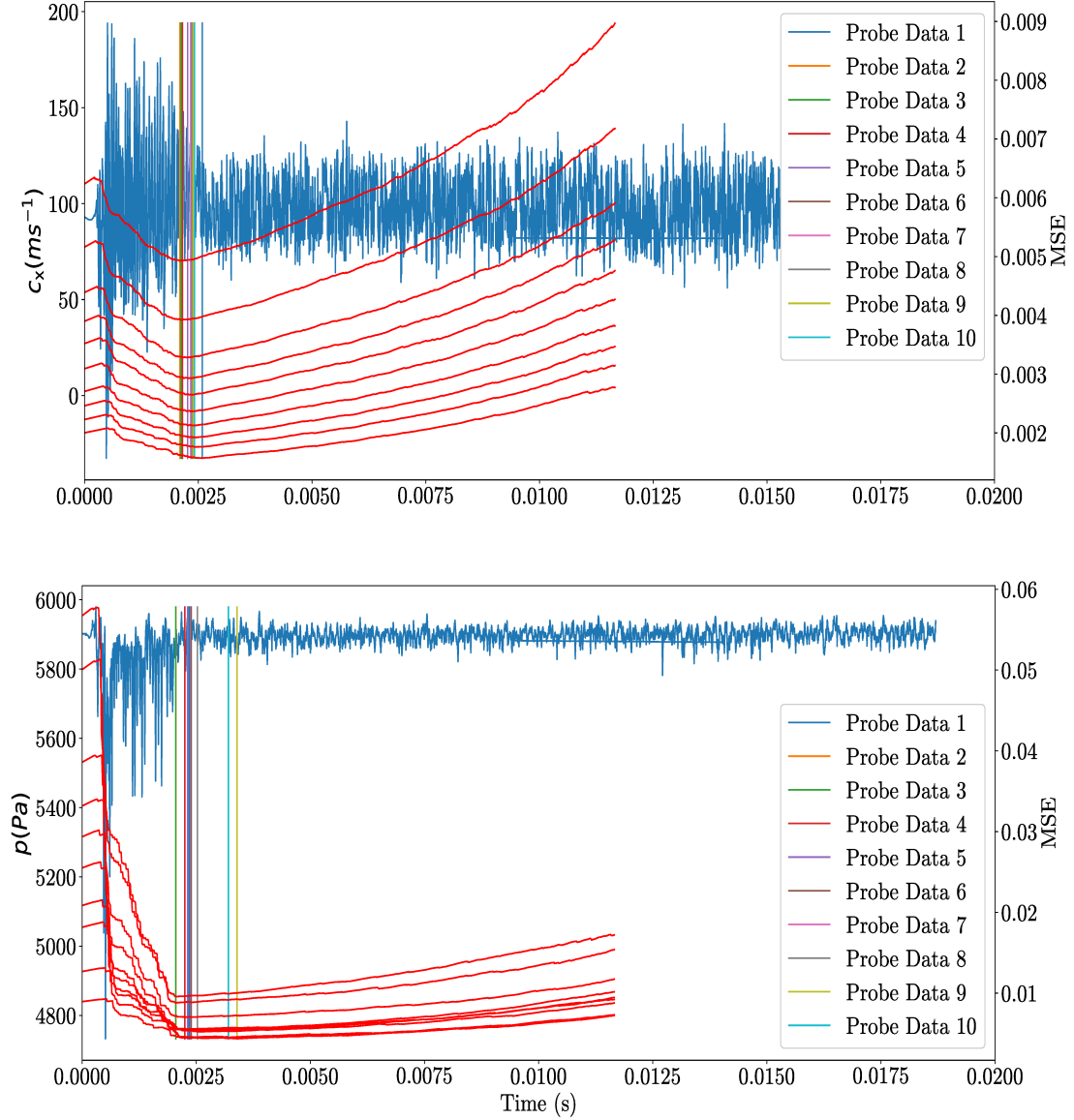


Figure 4.2: Monitoring the end of initial transient for axial velocity and pressure at different positions for HPC test case.

the turbulence and transition models, were performed using TRACE code (Becker et al. (2010), Edmund Kügeler (2005), Morsbach (2017) and Röber et al. (2005)). A detailed description of the methods implemented in TRACE can be found in the corresponding publications of the numerical methods department of the institute of propulsion technology. The code has been validated and successfully employed for many academic and turbomachinery topologies in previous studies, including Müller-Schindewolf (2022), Bergmann et al. (2022), Morsbach et al. (2020) and Morsbach et al. (2023). The aforementioned publication focus on high-fidelity turbomachinery applications.

4.5 Test case description: Low pressure turbine cascade

In this section, the LES results of the flow over the mid-span of the low-pressure turbine (LPT) cascade MTU-T161 are validated against experimental results (as already published in Fard Afshar et al. (2023a)). For this purpose, a mesh convergence study is performed, while evaluating the blade pressure distribution and wake flow losses. The highly loaded MTU-T161 was first investigated experimentally by Entlesberger et al. (2005) and published by Martinstetter (2010) and Ludewig et al. (2011). The MTU-T161 exhibits a very large Zweifel-number ($Zw = 1.2$) at the aerodynamic design point of $Re_{2,th} = 200,000$ and $M_{2,th} = 0.6$. The high Zweifel-number, which indicates a high blade loading, is accompanied by an intensive separation bubble over the entire measured Reynolds number range $70,000 \leq Re_{2,th} \leq 400,000$. The MTU-T161 cascade is characterized by a pronounced separation bubble with weak tendency to reattach at low Reynolds numbers. While such a configuration is not commonly used in industrial turbine designs, it serves as an ideal academic test case for studying transition, turbulence, and separation dynamics. The results obtained from this test case contribute to improving turbulence models and refining transition prediction methods for more industrially relevant applications. The characteristics of the aerodynamic design point of the MTU-T161 profile are given in Tab. 4.3.

Aerodynamic parameters	Values
$Re_{2,th}$	200,000
$M_{2,th}$	0.6
β_1	131.0°
β_2	25.0°
Pitch to chord	0.95
FSTI	4 %

Table 4.3: MTU-T161 profile classification of the aerodynamic design point

The profile boundary layer of MTU-T161 is subjected to a moderate turbulence intensity of $2\% \leq TI \leq 4\%$ from the free flow. A strong acceleration to $M_{is, \max} \approx 0.8$ takes place on the profile suction side up to $x/x_{lax} \approx 0.6$, followed by a strong deceleration. The deceleration causes the suction-side profile boundary layer to detach, which reattaches at the point where the laminar-turbulent transition is completed, due to an intensive momentum exchange in the free shear layer. In this respect, various transition-suppressing and transition-promoting mechanisms play a role at the mid-span flow of MTU-T161 cascade. These phenomena can be found in the two-dimensional core flow of the LPT. Depending on the operating point, the type of separation varies between short and long separation bubbles. All flow scenarios include a transition to turbulence before the reattachment point. In this work, the flow over the low-pressure turbine cascade is analyzed at $Re_{2,th} = 90,000$, $M_{2,th} = 0.6$. The selected operating point is an off-design point, in which the challenges of the RANS turbulence modeling will be highlighted, when compared to LES.

The experimental campaign of MTU-T161 cascade consists of seven blades with diverging side-walls. In order to capture the flow (even at the mid-span), the simulation model should be calculated as a three-dimensional model including side-walls. The blade to blade and spanwise view (including the diverging side-walls) of the LPT cascade is shown

in Fig. 4.3. Figure 4.4 shows the mid-span slice of the computational grid used for LES of MTU-T161 cascade, where every 4th grid point is shown.

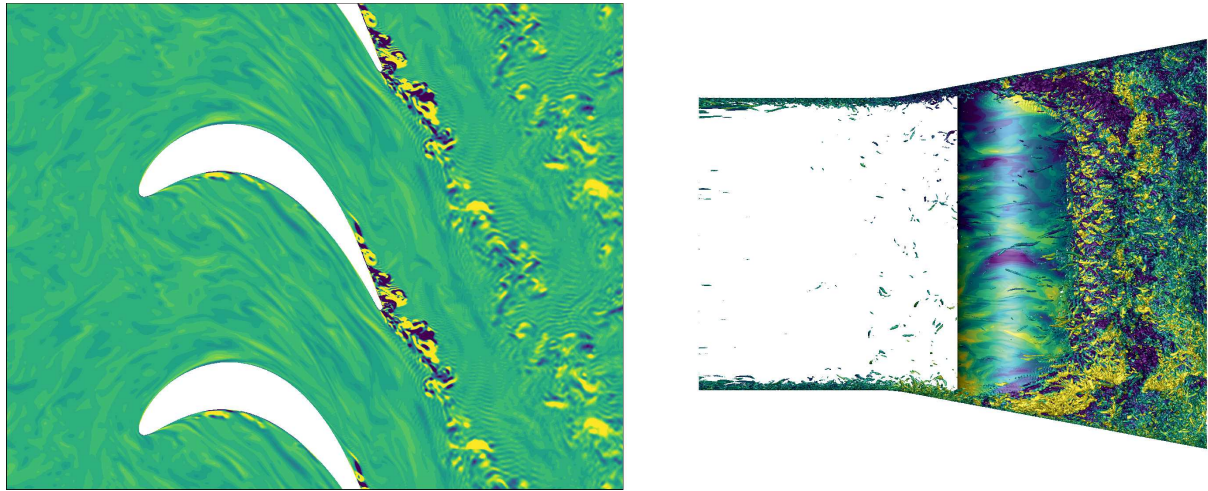


Figure 4.3: Blade to blade and spanwise view of MTU-T161 (left). Streamwise vorticity contour and spanwise velocity at mid-span of MTU-T161 are shown (right).

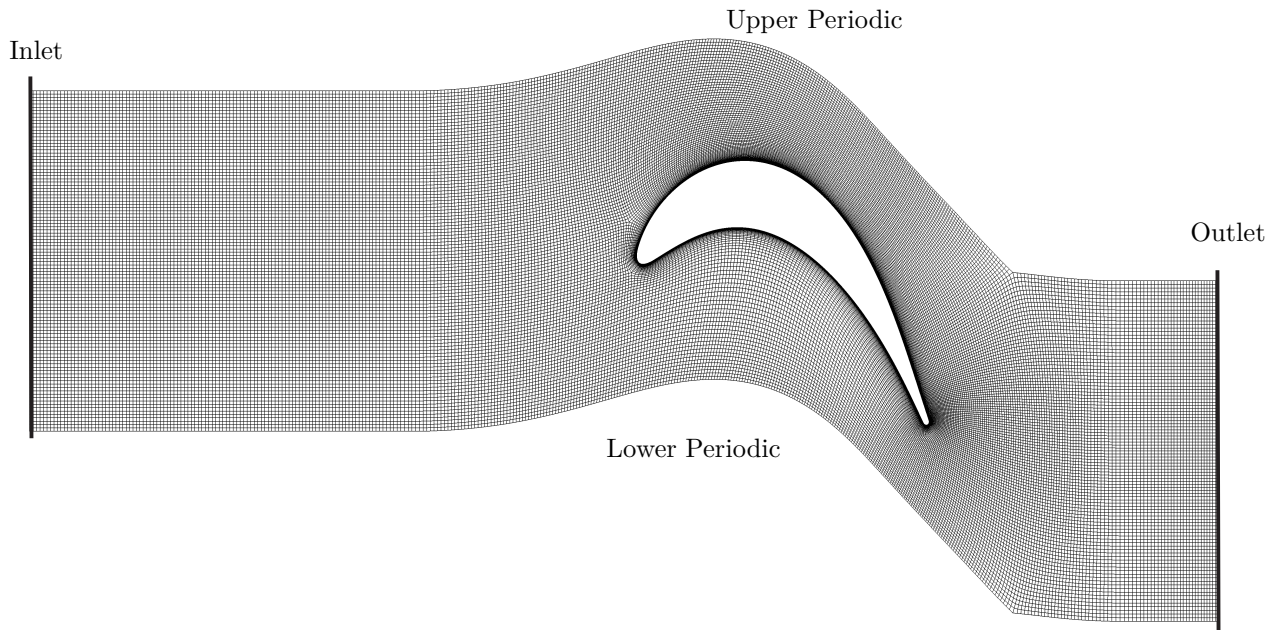


Figure 4.4: Two-dimensional slice of computational grid used for LES of MTU-T161 cascade. Every 4th grid point is shown.

The LES of the MTU-T161 cascade is initialized using a RANS solution. The RANS solution is used to iterate the boundary conditions in order to match the operating point. The LES are run for 40 convective time units to collect the (first and second order) statistics. In order to overcome the initial transient, the first 10 convective time units of the LES calculation are ignored. For the purpose of monitoring the convergence and initial transient, the methodologies introduced in Sec. D were employed. The convective time unit for the turbine cascade is defined with the chord length l divided by the outlet velocity magnitude ($t_c = l/c_2$). In the following section, the results of a series of LES are presented, analyzed

and compared against RANS results. The LES calculations of MTU-T161 are performed using the finite volume-LES (FV-LES) method (cf. Sec 4.1.1) and the RANS calculations using the $k - \omega$ -Viscous-Blending model (cf. Sec 2.5.3.3).

4.5.1 Validation

In this section, the validation of the numerical results, in terms of blade loading and flow losses, is presented. This validation demonstrates, that the results, analyzed in the following sections, include the proper quality, in terms of the spatial discretization resolution. In order to validate the numerical results at the blade mid-span, the isentropic Mach number (M_{is} cf. (4.6)) and skin-friction coefficient (c_f cf. (4.7)) distribution over the blade surface, are compared with the corresponding experimental data. The comparison enables the judgment, which resolution is needed to capture the correct blade loading (M_{is}), separation, transition and reattachment correctly. Additionally, the total pressure losses (ζ) in the mid-span wake region are compared with experimental data, to evaluate the accuracy of the LES results regarding the flow loss prediction. For this purpose, three LES simulations were performed using meshes with varying orders of accuracy. Table 4.4 provides details of the meshes used for validation, including the degrees of freedom, and the average and maximum non-dimensional cell sizes (in all three directions). The meshes are consequently refined in axial and spanwise direction. In the literature, there are many different ranges reported, regarding the non-dimensional cell sizes (cf. Sec. 4.1.3). Depending on the average non-dimensional cell size values, the finest and medium meshes (Mesh1 and Mesh2) meet the widely accepted criteria for wall resolution required for wall-resolved LES according to Georgiadis et al. (2010).

The time-averaged isentropic Mach number is defined as

$$\overline{M}_{is}(\mathbf{x}) = M_{is} = \sqrt{\frac{2}{\gamma - 1} \left(\left(\frac{\overline{p_{t,1}(t)}}{\overline{p(\mathbf{x}, t)}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right)}, \quad (4.6)$$

where $p_{t,1}(t)$ is the area-averaged total pressure taken from the inlet and $p(\mathbf{x}, t)$ is the static pressure on the blade surface. Consequently, the skin-friction coefficient ($\overline{c_f}$) is computed as following

$$\overline{c_f}(\mathbf{x}) = c_f = \text{sgn}(\mathbf{t}(\mathbf{x}) \cdot \overline{\boldsymbol{\tau}(\mathbf{x}, t)}) \frac{\|\overline{\boldsymbol{\tau}(\mathbf{x}, t)_W}\|}{p_{t,2}(t) - p_2(t)}, \quad (4.7)$$

where $p_{t,2}(t)$ and $p_2(t)$ are the area-averaged total and static pressures at the outlet, $\boldsymbol{\tau}(\mathbf{x}, t)_W$ is the wall shear stress vector and $\mathbf{t}(\mathbf{x})$ is the surface tangential vector orthogonal to the spanwise direction pointing in positive axial direction.

Figure 4.5 illustrates the normalized isentropic Mach number (M_{is}) distribution and skin-friction coefficient (c_f), as described in (4.6) and (4.7), for three meshes with different degrees of freedom. The results of the Mesh1 (fine) and Mesh2 (medium) show almost no differences in both M_{is} and skin-friction distribution along the blade. Both meshes align very well with the experimental results in terms of pressure distribution (e.g. M_{is}), with slight deviations observed on the first half of the suction side. However, these deviations

Mesh	nDOF (10^6)	Maximum			Average		
		Δx^+	Δy^+	Δz^+	Δx^+	Δy^+	Δz^+
Mesh3	28	65	1	60	35	0.6	30
Mesh2	90	50	1	40	30	0.5	20
Mesh1	146	35	1	25	15	0.5	10

Table 4.4: Discretization properties of the LES simulations for MTU-T161 cascade

(of Mesh1 and Mesh2) remain within the experimental error margin, indicated by the symbol size. Both meshes show similar blade loading, separation and transition behavior. The reason for the small deviation between Mesh1, Mesh2, and measurements on the front part of the suction side could not be found in the numerical simulations. One hypothesis is the undersampling of measurement probes in this region, which also matches the good prediction of the under-resolved Mesh (Mesh3) with the measurement results in this region.

On the other hand, Mesh3 (coarse mesh) shows slightly better prediction of the experimental isentropic Mach number distribution in the first half of the suction side compared to other mesh resolutions. Nevertheless, Mesh3 predicts an earlier separation and onset of transition, on the suction side, compared to finer meshes. The differences on the aft part of the profile suction side, regarding the separation, transition and reattachment, are clearly visible in the skin-friction distribution (bottom part of Fig. 4.5). In case of under-resolution of high-fidelity simulations (Mesh3 in our case), it is a typical behavior that flow separation and transition happen earlier (than high resolution meshes), resulting in larger separation bubbles. Low resolution meshes are not able to resolve the pre-transitional fluctuations as correctly as high resolution meshes, therefore showing the tendency to separate earlier. On the pressure side, Mesh3 shows a smaller separation bubble, than the other two meshes.

Another important factor while designing new blade profiles is the total pressure loss in the wake region, which is computed as follows

$$\bar{\zeta}(y) = \zeta = 1 - \frac{\overline{p_t(y, t)}}{\overline{p_{t,1}(t)}}, \quad (4.8)$$

where $p_t(y, t)$ is the total pressure at the pitchwise position (y) in the wake region and $p_{t,1}(t)$ is the area-averaged total pressure taken from the inlet.

Figure 4.6 illustrates the normalized total pressure loss in the wake region ($x/x_{lax} = 140\%$), for the three meshes. Comparing the results of Mesh1 and Mesh2, only very marginal difference between these two meshes on the pressure side branch of the wake flow loss distribution are observed. Mesh3 predicts smaller wake flow loss peak and width, compared to the other two meshes. Mesh3 predicts 8% smaller integral flow loss values than the other two meshes. Additionally, the peak value of the wake flow loss predicted by Mesh3 differs from the other two meshes by 2%.

The findings in this section show that Mesh2 is fine enough to represent both the isentropic Mach number over the blade and the total pressure flow loss in the wake according to experiments. Therefore, for the remaining parts of this chapter, the results of Mesh2

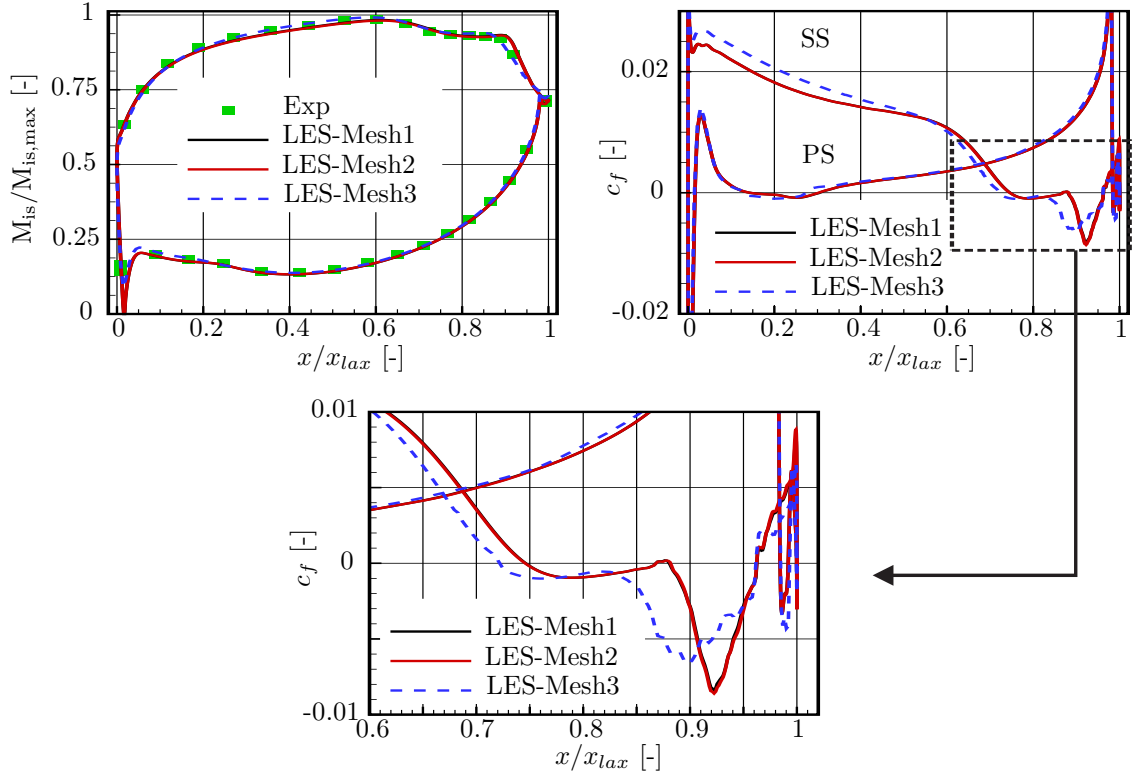


Figure 4.5: Top: Normalized isentropic Mach number and skin-friction coefficient distribution at mid-span of MTU-T161 by different LES meshes. Bottom: Zoomed view of the skin-friction coefficient distribution at mid-span of MTU-T161 for $Re = 90,000$ by different LES meshes.

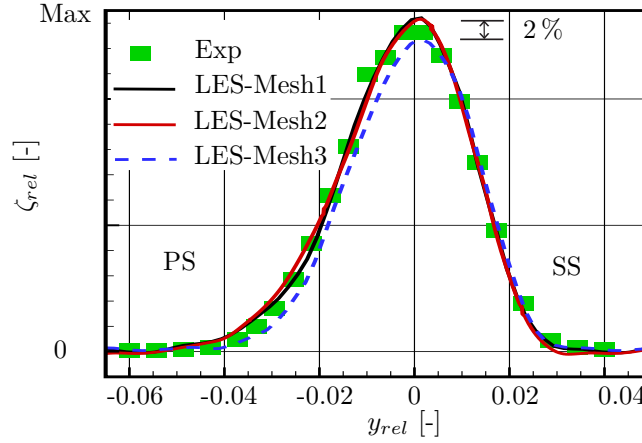


Figure 4.6: Normalized total pressure loss in wake region ($x/x_{lax} = 140\%$) at mid-span of MTU-T161 by three different LES meshes

are used for the analysis. A comparison of Mesh1 and Mesh2 in terms of second-order statistics, such as Reynolds stresses in the wake, revealed no notable differences (not shown here).

Aerodynamic parameters	Values
Re_{in}	492,000
M_{in}	0.4
Aspect ratio	1.25
FSTI	2.5%

Table 4.5: Geometric and aerodynamic characteristics of the high-pressure compressor cascade

4.6 Test case description: High pressure compressor cascade

The test case investigated in this section is a high-pressure high-subsonic compressor cascade with the aerodynamic design point and geometrical characteristics shown in Tab. 4.5. The analyzed compressor cascade corresponds to the center section of a modern high-pressure compressor rotor and has been analyzed experimentally by Winter (2016), Hösen (2019), and Lang (2024). Because of the low aspect ratio of this profile the flow at the mid-span is highly influenced by secondary flow. Therefore, in order to capture the flow (even at the mid-span) the simulation model is calculated as a three-dimensional model including side-walls. The flow over the high-pressure compressor cascade is analyzed at $Re_{1,th} = 492,000$, $M_{1,th} = 0.4$. The selected operating point (inflow angle) is an off-design point, in which the challenges of the RANS turbulence modeling will be highlighted when compared to iLES. The computational challenges of calculating the flow over the high-pressure compressor cascade, investigated in this chapter, are the higher Reynolds number (compared to the low-pressure turbine cascade) and the necessity of including the side-walls (as described before). In order to tackle the computational challenges of this numerical test case, the high-order discontinuous Galerkin spectral element iLES method (instead of FV-LES) will be used, and its computational benefits, as described in 2.5.2.2, will be utilized.

The iLES of the high-pressure compressor cascade is initialized by a RANS solution, which is used to iterate the boundary conditions in order to match the operating point. The iLES is run for 20 convective time units to collect the (first and second order) statistics. In order to overcome the initial transient, the first 5 convective time units of the iLES calculation are ignored. The convective time unit for the compressor cascade is defined with the chord length l divided by the inlet velocity magnitude c . The blade-to-blade and spanwise view (including the side-walls) of the high-pressure compressor cascade is shown in Fig. 4.7.

4.6.1 Validation

To validate the numerical results, the isentropic Mach number distribution over the blade surface and the total pressure loss in the wake are compared with experimental data at blade mid-span plane. For this purpose, three iLES were performed using meshes with varying orders of accuracy. As already described in 2.5.2.2, the approach, known as

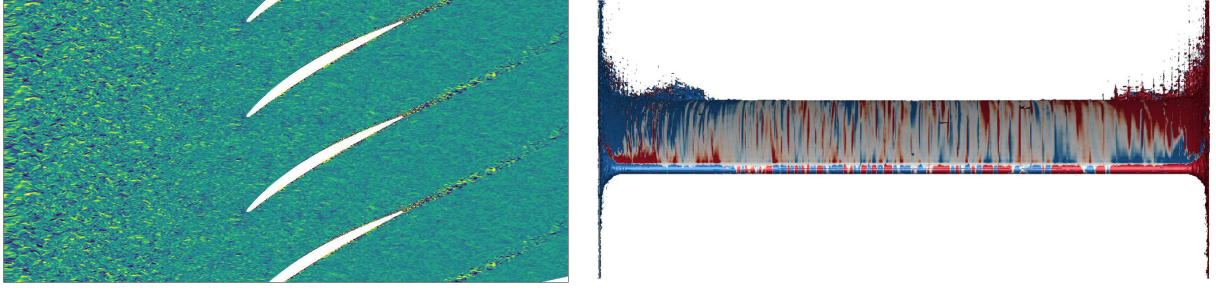


Figure 4.7: Left: Streamwise vorticity contour in blade to blade view. Right: Q-criteria coloured with streamwise vorticity in spanwise view of the high-pressure compressor.

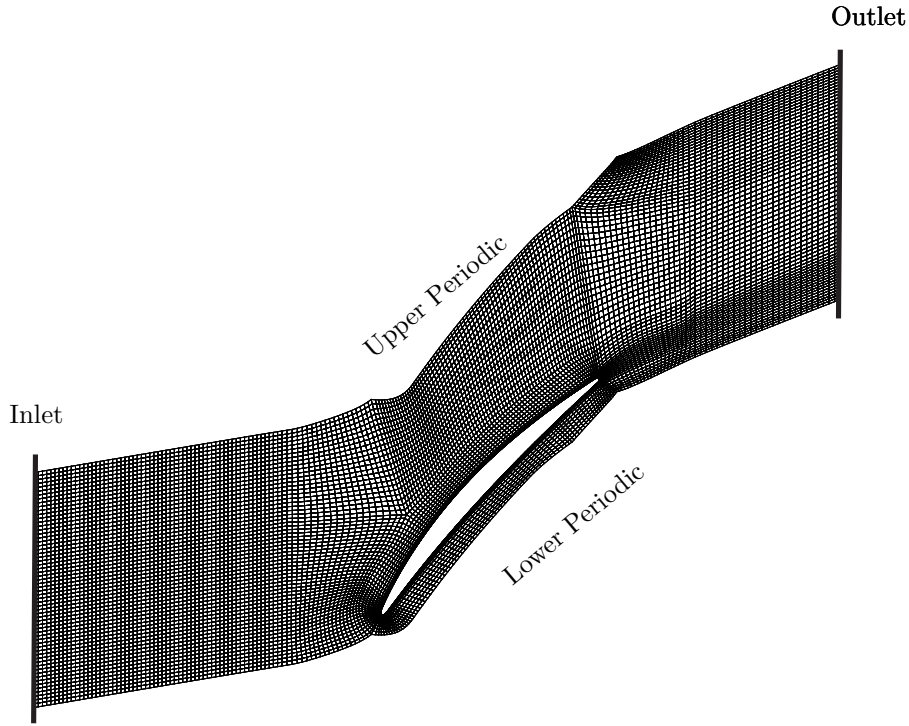


Figure 4.8: Two-dimensional slice of computational grid used for iLES of the compressor cascade. For clarity, only the high-order grid cells are displayed, while the 25 solution nodes contained in each two-dimensional element are omitted.

order refinement or polynomial-refinement (p-refinement), globally refines the spatial discretization (by increasing the polynomial order) without requiring local mesh refinement. Table 4.6 provides the details of the meshes used for the validation, including the order of accuracy, resulting mesh characteristics e.g., number of degrees of freedom (nDoF) and the average and maximum non-dimensional cell sizes, as defined in 2.5.2.2. The maximum values of non-dimensional cell sizes are mostly concentrated around the leading edge. By comparing the values of Tab. 4.6 with the results presented in literature (cf. Tab. 4.2), the spatial discretization used for this thesis can be categorized between medium and high resolution quality.

Polynomial order (p)	nDoF (10^6)	Maximum			Average		
		Δx^+	Δy^+	Δz^+	Δx^+	Δy^+	Δz^+
6	422	40	3.8	95	24	0.9	24
5	266	47	4.4	112	28	1.1	29
2	34	103	8.2	207	44	1.9	48

Table 4.6: Discretization properties of the iLES simulations for the high pressure compressor cascade. nDoF refers to the number of degrees of freedom. Polynomial order (p) refers to the order of the element-local polynomial approximation of the solution. The non-dimensional cell sizes are normalized by the polynomial degree of the solution $p+1$.

Figure 4.9 illustrates the normalized isentropic Mach number (M_{is}) distributions and wake flow losses for three degrees of freedom at the blade mid-span plane. The results of the meshes $p = 5$ and $p = 6$ show only marginal differences in both M_{is} distribution along the blade surface and total pressure loss in the wake. Both meshes align very well with the experimental results in terms of pressure distribution (e.g., M_{is}), with slight deviations observed on the second half of the pressure side. However, the deviations of $p = 5$ and $p = 6$ remain within the experimental error margin, indicated by the error bars. The numerical results of $p = 5$ and $p = 6$ predict a separation bubble on the suction side, which initiates the boundary layer transition. This boundary layer transition can be identified as a distinct pressure jump on the suction side. However, the mean streamlines of the numerical results (not shown here) do not indicate a definitive mean flow separation, suggesting that the flow remains largely attached. The boundary layer appears to be on the verge of separation due to the adverse pressure gradient. The velocity profile in this region of the boundary layer exhibits a turning point, signifying a strong positive pressure gradient. This condition places the boundary layer in a critical state, where minor changes in flow parameters could lead to significant changes in transition phenomena. The boundary layer in this critical state might exhibit localized turbulence spots (laminar-turbulent transition points), meaning the flow is in the early stages of turbulence but has not fully transitioned yet. The experimental results show only a tendency toward separation bubble due to the limited resolution of the measurements.

In contrast, the results of the lowest-resolution mesh ($p = 2$) deviate significantly from both the finer meshes and measurement results. The $p = 2$ mesh fails to predict a separation bubble on the suction side, due to its lower discretization order. This behavior is a well-known issue for under-resolved meshes, which struggle to capture the transition mechanism and profile pressure distribution accurately. Additional discrepancies are evident, between the $p = 2$ mesh and the other two meshes near the leading edge on both the pressure and suction sides and along the pressure side near the trailing edge, further highlighting the limitations of the low-order discretization.

In the wake region, the lowest mesh order ($p = 2$) significantly underestimates the total pressure peak loss (compared to the measurements and the other two meshes). This underprediction persists across all wake planes, with the discrepancy between mesh orders increasing further downstream. The observed underestimation is attributed to higher numerical dissipation associated with the lower spatial discretization in the wake region. Additionally, the $p = 2$ mesh predicts a broader wake flow loss distribution, with the

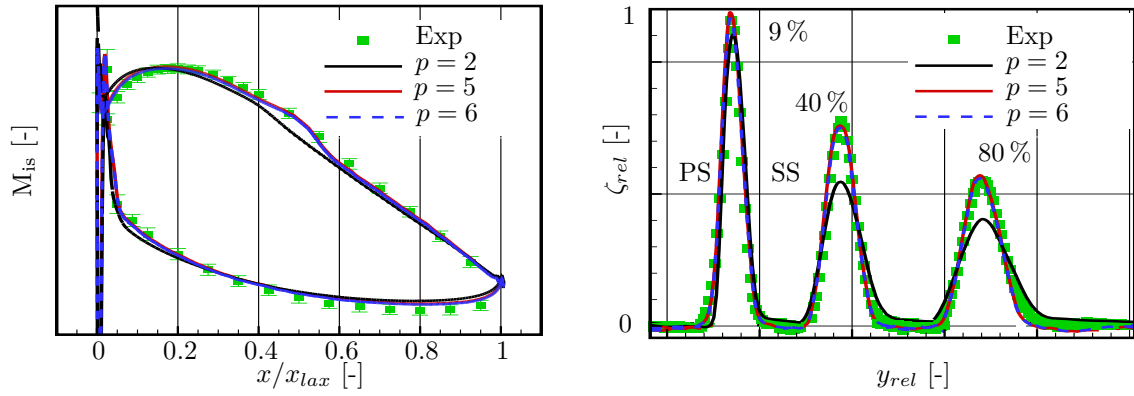


Figure 4.9: Normalized isentropic Mach number distribution (left) and normalized total pressure loss in wake (right) by iLES for three discretization orders. The measurement results include error bars. The wake region is shown for three axial planes.

width continually increasing downstream. In contrast, the results for $p = 5$ and $p = 6$ show negligible differences across all wake planes. Both meshes accurately capture the peak flow loss and the width of the wake distribution, aligning closely with experimental measurements. The integral flow loss deficit between the low order mesh and the other two meshes varies between 8 % and 14 % for different axial wake planes. This significant discrepancy highlights the limitations of lower-order spatial discretization in accurately capturing the wake flow loss characteristics. Additionally, these findings confirm that the $p = 5$ mesh is sufficiently refined to represent both the experimental isentropic Mach number over the blade and the experimental total pressure loss in the wake. Therefore, for the remaining parts of this chapter, the results of $p = 5$ are used for the analysis.

Beyond spatial accuracy validation, additional assessments, such as convergence studies and the accurate prescription of boundary conditions (as presented in Sec. 4.2.1 and Sec. 4.4), were conducted. However, these validations are not presented here.

5 Numerical Results of a Highly Loaded Low Pressure Turbine Cascade

The numerical results in this chapter were already validated against measurements in Sec. 4.5. These results represent one of the first numerical analyses of this low-pressure turbine cascade conducted using high-fidelity (LES) methods. Additionally, the LES results are compared against RANS results to examine the benefits of using the LES approach over the standard RANS models. Furthermore, the turbulent behavior of the low-pressure turbine flow will be investigated in terms of anisotropy, entropy, spectral, modal, and quadrant analysis. The last section of this chapter deals with the influence of the free-stream turbulence on the boundary layer and wake characteristics.

5.1 Time-averaged results at mid-span

In this section, the results of the LES-Mesh (selected in the last section) are compared against the best-practice RANS results. The focus of this section is the comparison of the two methods, in terms of blade surface values, wake flow losses, and turbulence analysis at mid-span of the MTU-T161 profile.

5.1.1 Surface analysis at mid-span

The results of LES-Mesh2 (validated in Sec. 4.5.1) are compared against the $k - \omega$ -VB-model, regarding the prediction of M_{is} over the blade and total pressure losses in the wake. $k - \omega$ -VB-model was chosen as the best-practice RANS setup in preliminary numerical studies, involving turbulence model and CFD setup configuration variations. For transition modeling, the $\gamma - Re_\theta$ Menter transition model (Langtry et al. (2009)) is employed. Additionally, the stagnation point anomaly extension by Kato et al. (1993) is used.

Blade results

Figure 5.1 shows the isentropic Mach number (M_{is}) distribution at mid-span of the MTU-T161 profile for RANS and LES methods. On the suction side, the M_{is} -distribution starts to deviate between the methods, at $x/x_{lax} = 20\%$, where the RANS-simulation leads to larger isentropic Mach number and a shift of the suction peak downstream of the LES peak, from $x/x_{lax} = 62\%$ to 65% . This difference leads to a different prediction of the blade loading by each method. Looking at the M_{is} and c_f distribution, it is observed that RANS predicts a later start ($x/x_{lax} = 78\%$) of the flow separation than LES. Additionally, the RANS result predicts a sooner flow reattachment than LES before the trailing edge, at $x/x_{lax} = 93\%$, and consequently a smaller separation bubble than LES. On the pressure

side, RANS shows a larger separation bubble than LES, extending from $x/x_{lax} = 10\%$ to 45% . The main differences between the methods (RANS and LES) are the prediction of the separation bubbles (on the suction and pressure surfaces) and the resulting blade loading.

Figure 5.1 shows the time-averaged skin-friction coefficient distribution over the MTU-

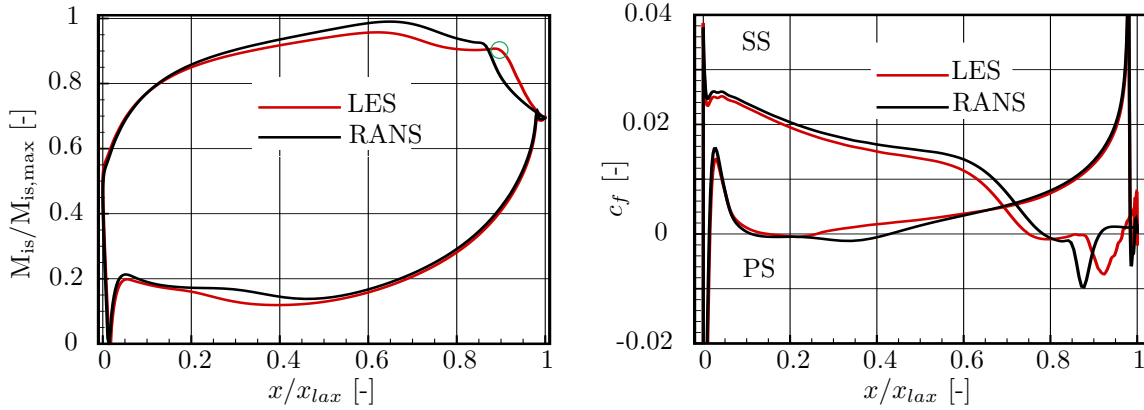


Figure 5.1: Normalized isentropic Mach number (left) and skin-friction coefficient (right) distribution at mid-span of MTU-T161 by RANS and LES

T161 profile for RANS and LES methods. The time-averaged skin-friction coefficient confirms the M_{is} -distribution and allows a more precise determination of the separation and reattachment points ($\overline{c_f} = 0$). A specific characteristic of the LES result is a small area of positive wall shear stress between the separation and reattachment points (between $x/x_{lax} = 85\%$ and 88%). This area results from a very small additional vortex core (compared to the main separation bubble). The small vortex core exists between the main separation bubble and the blade surface and has an opposite circulation to that of the main separation bubble. Another indication of the small vortex is the rapid increase of the isentropic Mach number before the recovery in LES results (shown as green circle in Fig. 5.1).

The comparison of the surface values between LES and RANS results, shows the differences of each model in predicting the separation bubble and the different flow behavior on suction and pressure side. In summary, RANS predicts a smaller separation bubble on the suction side but a bigger one on the pressure side than LES. The deficiencies of the RANS model to accurately predict the pressure distribution may be tracked back to the inadequate reproduction of the turbulence effects in the transitional and separation region. Since the separation point is in the laminar and transitional state, the turbulence is mostly generated at the outer part of the separation bubble, where strongly anisotropic characteristics are present, as will be shown later. These characteristics are also propagated into the wake. In order to analyze the aforementioned hypothesis, the turbulence behavior at the mid-span section of the MTU-T161 will be examined for both LES and RANS in the next section. The TKE analysis in the next sections will be presented as contour plots in the next sections, since it is not possible to conduct such comprehensive analyses via measurements. This emphasizes the advantages of high-fidelity simulations. Additionally, the emphasis will be predominantly directed towards the suction side flow. Although the flow on the pressure side exhibits interesting characteristics.

5.1.2 TKE analysis at mid-span

Figure 5.2 shows the normalized TKE and TKE-production for both RANS (modeled) and LES (resolved). The modeling content of the turbulence production in LES is very negligible, therefore only the resolved part is shown. The results are normalized either based on the square of exit velocity or the maximum values of the TKE-production (P_{TKE}) of both LES and RANS results.

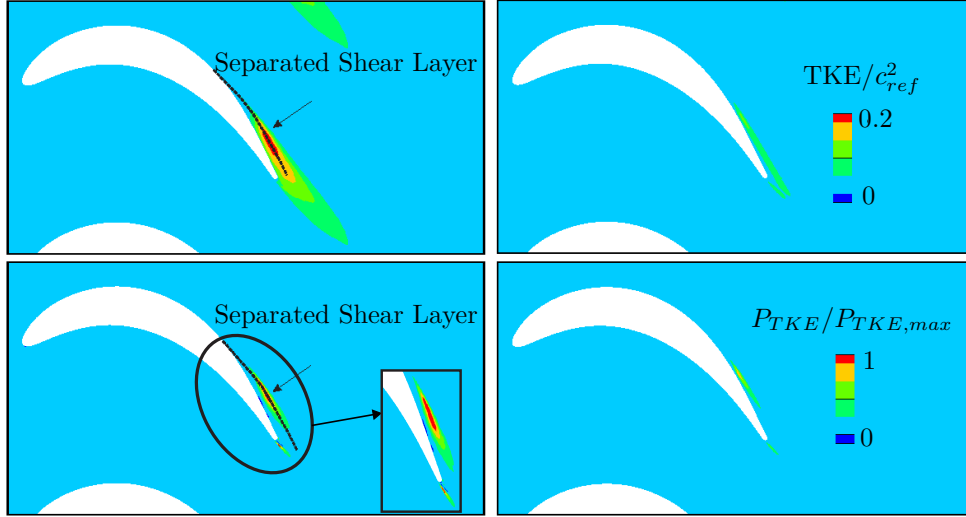


Figure 5.2: Normalized TKE cf. (2.4) (top) and normalized TKE-production cf. (2.43) (bottom) for LES (left) and RANS (right) at mid-span of MTU-T161. The separated shear layer is shown as a dashed line over the suction side.

Looking at the normalized TKE-contour, different distributions and ranges of TKE values between RANS and LES are observed. The LES result shows the highest TKE values over the separation bubble along the separated shear layer (along the dashed line). The TKE content is convected downstream into the wake region, while decreasing in intensity. The values of the modeled TKE by RANS are, in general, lower than the LES values over the whole suction side. The RANS result is in a more elongated shape and closer to the blade. The different distribution of TKE over the suction surface, directly affects the separation bubble behavior and as already shown in Fig. 5.1, results in a smaller separation bubble, predicted by RANS, compared to LES. Similarly, the TKE-production contour of LES shows the highest values of turbulence production over the separation bubble and near trailing edge. The blue regions near the wall (on the suction side surface), identify the corresponding negative values of TKE-production. In turbulence, energy typically cascades from large scales to smaller scales (forward cascade), but in some regions, especially near walls or in shear layers, energy can be transferred from small scales back to larger scales (inverse cascade), leading to negative production. The distribution of TKE-production is more discrete along the separated shear layer, which highlights the region where high levels of TKE is produced.

The TKE-production predicted by RANS, differs from LES in terms of distribution and level. The TKE-production contours reveal that the separation bubble predicted by RANS, is smaller than LES in wall-normal direction. Additionally, the amount of TKE predicted by LES is considerably higher than RANS (up to 1.5 times higher). These contour plots

highlight the different spatial prediction of turbulence by each method, with a focus on the separation bubble and the near trailing edge region.

After the TKE behavior has been examined qualitatively, the evolution of TKE will be compared in a more quantitative manner using both methods. For this purpose, Fig. 5.3 shows the normalized TKE-distribution normal to the suction side surface for RANS (modeled) and LES (resolved). The RANS results are shown as dashed line and LES results as solid lines. The normalized TKE values are plotted against non-dimensionalized wall distance. Starting from the leading edge, noticeable differences between RANS and

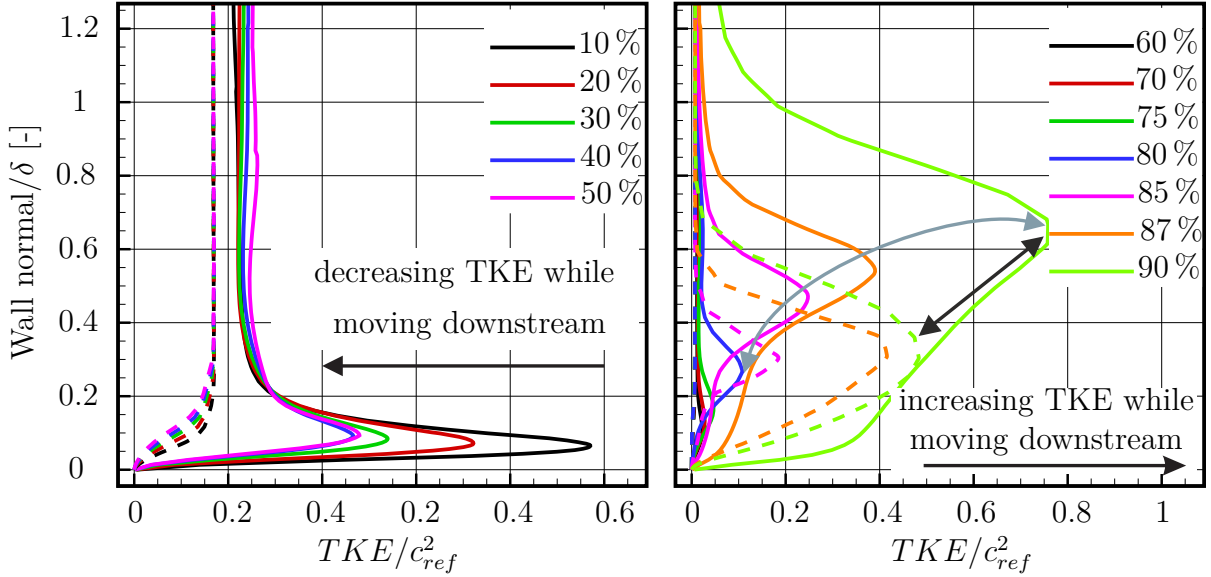


Figure 5.3: Evolution of normalized TKE over the suction side by RANS (dashed) and LES (solid) at mid-span of MTU-T161. Left: First half of the suction side. Right: Second half of the suction side

LES regarding the evolution of TKE, are observed. The LES result shows a single, distinct TKE-peak in the boundary layer which decreases continuously in the amplitude while approaching $x/x_{lax} = 50\%$. RANS result shows the same trend but the TKE values are approximately 4 times smaller than LES. Additionally, RANS result does not show a distinct peak near the wall, as in LES. The limited TKE evolution of the RANS method is directly related to the modeling concept of $k - \omega$ -VB Model as described in 2.5.3.3. $k - \omega$ -VB Model addresses the deficiencies of conventional RANS closures in predicting transition in low-pressure turbine flows by modifying the turbulence production terms such that eddy viscosity is damped in laminar or weakly transitional regions, thereby suppressing the spurious overproduction of TKE that is typically observed in separated shear layers.

In the second half of the profile suction side (right side of Fig. 5.3), according to LES, the TKE values continuously rise while moving further downstream along the suction side (from $x/x_{lax} = 60\%$ to $x/x_{lax} = 90\%$). At the same time the TKE peaks move away from the wall, while continuing along the suction side surface (grey arrow). RANS shows the same trend as LES, but at all boundary layer cuts, RANS predicts lower TKE values than LES. The difference of the wall normal position of TKE-peaks, between RANS and LES, is related to the different separation bubble sizes predicted by these two methods (black

arrow). The difference between the TKE-peak values (RANS vs. LES) is considerably smaller in the second half of the suction side (right figure). This behavior is consistent with the difficulties of RANS models in predicting the correct TKE values in the laminar and transitional flow regimes (first half), but performing better after the separation and entering the fully turbulent flow regimes (second half). This shortcoming is because most turbulence models and their modifications are calibrated for fully turbulent flow scenarios.

In a broader sense, the discrepancy between RANS and LES arises because standard eddy viscosity models fail to generate the right amount of turbulence intensity in transitional and pre-transitional flows, leading to an over- or under-estimation of laminar regions. After separation and reattachment, where the flow becomes fully turbulent, the TKE difference between RANS and LES decreases and RANS performs better in capturing fully developed turbulence. The influence of pressure gradients on the redistribution of Reynolds stresses can be deduced from their impact on the mean flow field. An adverse pressure gradient has been shown to decrease wall shear stress and induce the formation of shear layers farther away from the wall within the boundary layer. Consequently, the production of turbulence is displaced from the near-wall region toward the outer portion of the boundary layer. As a result, the near-wall peak of turbulent kinetic energy is diminished, while a broader maximum emerges in the outer region of the layer, as shown by Durbin et al. (2010).

After the design and turbulence values at the mid-span section of the suction side have been examined, the time-averaged wake values will be analyzed in more detail in the next section

5.1.3 Total pressure loss at wake plane

Figure 5.4 shows the total pressure loss (as described in (2.94)) in the wake plane, $x/x_{lax} = 140\%$, computed via LES and RANS. The total pressure loss peak is predicted similarly by both methods. However, the shape of the total pressure loss is predicted considerably different by both methods. The RANS turbulence model used in this work, shows difficulties in predicting the correct boundary layer at the trailing edge and turbulence spreading in the wake area, which results in a more narrow wake flow loss shape than LES. This is a well-known deficit related to RANS turbulence models. The difference in the wake flow loss shape between these two methods results in an under-prediction of the integral wake flow loss values about 32 % by the RANS method. The very good accuracy of the LES result in predicting the wake flow loss, compared to the measurements, was already demonstrated in Fig. 4.6. At the same time, the discrepancy between RANS and LES results shows the shortcomings of the RANS method (used in this work) to predict the flow losses, and highlights the importance of high-fidelity methods in correctly predicting the design variables.

Looking at the turbulence production at the same wake plane, there is a remarkable difference between LES and RANS. The RANS model is not able to produce the same TKE level as LES. There is a deficit of up to 2.5 times between RANS and LES regarding the turbulence production peaks. It should be mentioned again that the resolved turbulence production in LES is being compared with the modeled turbulence production in RANS.

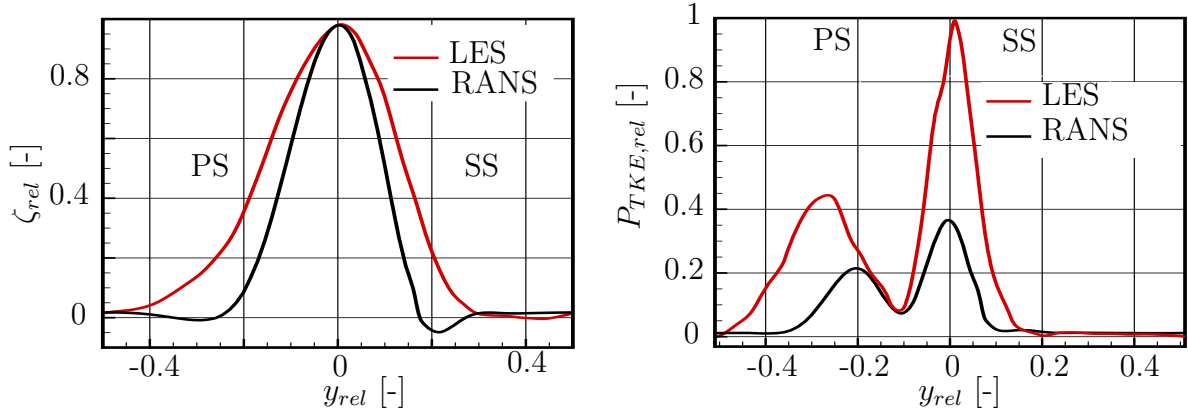


Figure 5.4: Normalized total pressure loss and normalized TKE production in wake region ($x/x_{lax} = 140\%$) at mid-span of MTU-T161 by RANS and LES

Nevertheless, looking at the shape of the turbulence production peaks, both LES and RANS are very similar. Both methods show a higher turbulence production peak related to the suction side flow. The correct prediction of turbulence production peaks via the RANS model is a positive aspect, which is related to the turbulence modeling extensions introduced in Sec. 2.5.3.3. The discrepancies in the level of the turbulence production between LES and RANS remains a crucial deficit of the corresponding RANS turbulence model. As already shown before, the discrepancies regarding the prediction of TKE values, start further upstream on the suction side and extend to the wake region. As will be shown later, the discrepancy in predicting the turbulence production, has a major influence on the prediction of the flow losses in the wake region.

5.2 Further Analysis of LES results

In order to better understand the results and discrepancies between RANS and LES, the turbulence behavior of the LES results at the mid-span plane is analyzed in further detail in the current chapter. Analyzing the TKE, showed the differences between the two methods, in terms of normal Reynolds stresses. Nevertheless, the role of the Reynolds shear stresses in producing turbulence is as important as normal stresses. In the following section, the significance of anisotropy in the composition of turbulence will be emphasized.

5.2.1 Anisotropy

Following the distribution and evolution of TKE, as predicted by RANS and LES at the midspan of the MTU-T161 cascade, the turbulence states of RANS and LES will be examined. To get a better overall understanding of the turbulence state, Fig. 5.5 shows the componentality contours of the Reynolds stress anisotropy tensor in mid-span plane. To do this, the methodology of Emory et al. (2014) is used, in which the weights C_{ic} of the barycentric map are mapped to the RGB channels (cf. Sec. 2.2.1). Measuring and analyzing the anisotropy is a way to quantify the effect of the shear on a given range of scales in a flow.

First, the LES result, shown on the left side of Fig. 5.5, will be examined. The inflow is characterized by moderate isotropic three-component (3C) incoming turbulence, which is consistent with the applied turbulent boundary conditions (region 1). In free-stream, the isotropic turbulence state enters the passage until it reaches the acceleration region, promoting a shift to two-component (2C) turbulence indicated by the change from blue to green color in Fig. 5.5 (region 2). In this region, spanwise ($\overline{c'_3 c'_3}$) and pitchwise ($\overline{c'_2 c'_2}$) fluctuations make an almost equal contribution to the turbulence level, developing a 2C turbulence structure. The origin of the 2C turbulence state can be mainly traced back to the velocity gradient governing in the passage flow. Following a streamline in the middle of the passage, as shown in the Fig. 5.5, the flow experiences a significant velocity gradient along and normal to the streamline. The high pitchwise gradient promotes the production of turbulent energy in pitchwise direction. The energy is partially redistributed to the other two components. The streamwise gradient, in an identical fashion, causes a partial redistribution to the pitchwise and spanwise component.

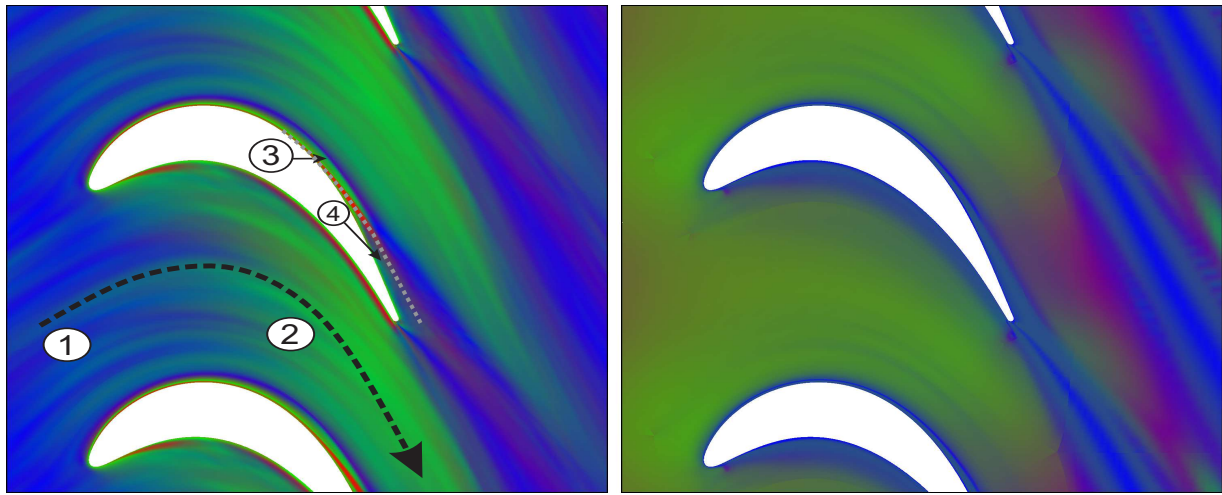


Figure 5.5: Componentality of Reynolds stress anisotropy tensor for LES (left) and RANS (right) at mid-span of MTU-T161. The colors of the contours are described in 2.2.1.

The near wall area of the profile is dominated by one-component (1C) turbulent state, in this case streamwise Reynolds stress ($\overline{c'_1 c'_1}$). The 1C layer (red color) spans from near leading edge to the separation point. After the flow starts to separate, the prominent 1C turbulence state is stretched along the separated shear line, indicating the maximum shear above the reverse flow region (grey dashed line). This behavior was also observed and noticed in the TKE distribution contour (Bottom row of Fig. 5.2). Since the turbulence production term of the Reynolds stress equation (2.35) includes the velocity gradient e.g. shear of the flow, it can be concluded that high shear causes the turbulence structures to elongate and align with the direction of the mean flow. Therefore, there are larger streamwise fluctuations compared to the other components and the energy is transferred from the mean flow mostly to the streamwise turbulence, creating the 1C turbulence state.

The second region of interest is the 2C area directly above the 1C layer next to the wall (region 3). The 2C layer follows the same path as the 1C near the wall. The origin of this phenomenon can be traced back to the redistribution effect of the incoming free-stream

3C isotropic background turbulence at the stagnation point of the leading edge. As has been shown by Xiong et al. (2007), the wall normal fluctuations at the stagnation point decrease close to the wall due to the non-penetration condition. The energy of the wall normal fluctuations is transferred to the other two components, creating a 2C state in the vicinity of the stagnation point. This state is convected downstream, with the mean flow tangential to the blade surface, causing the 2C layer (green color). The same behavior holds for the turbulence entrainment from the free-stream into the laminar boundary layer. In this case, the free-stream turbulence enters the laminar boundary layer and the laminar boundary layer patch undergoes the same (as at the leading edge) redistribution of the wall-normal component, together with the turbulence damping. As a result, the 2C layer is still present along the complete laminar boundary layer part. In the separated region (downstream of region 3), the 2C turbulence is dominant close to the wall, while the turbulence away from the wall becomes isotropic. The area inside the separation bubble is characterized by a mixture of 2C and 3C turbulence (region 4).

The RANS results exhibit a markedly different turbulence anisotropy behavior compared to LES. In the passage flow, RANS predicts predominantly two-component (2C) turbulence (green regions), in contrast to the LES results, which show 2C turbulence only after the flow enters the acceleration region. In the boundary layer, LES captures a more complex anisotropy distribution, while RANS uniformly predicts a three-component (3C) turbulence state (blue regions). This behavior closely mirrors the TKE distribution shown in Fig. 5.2. In the wake region, both methods show qualitatively similar anisotropy patterns, with turbulence primarily exhibiting a 3C character. However, the extent of this region differs. RANS predicts a narrower wake of 3C turbulence compared to LES. Moreover, RANS does not capture the secondary stream of 3C turbulence (blue color stream) generated by the separated shear layer, which is clearly observed in the LES results. Additionally, there is a blending region (3C and 1C) predicted by LES in the wake region, which is not visible in the RANS result. On the pressure side, significant differences are also evident in the predicted turbulence anisotropy within the boundary layer. Similar to the suction side, RANS predicts a fully 3C turbulent boundary layer. In contrast, LES reveals a more detailed structure, with both 1C and 2C turbulence. The 1C regions are primarily associated with the separation bubble and the detached shear layer, while the near-wall region is dominated by 2C turbulence. The results above were already presented and discussed in Fard Afshar et al. (2023a), while also analyzing the wall normal distribution of Reynolds stresses along the suction surface.

In general, a very different turbulence anisotropy behavior is predicted by RANS than LES, which is directly related to the deficiencies in modeling assumptions of the RANS turbulence models, especially when using a linear eddy viscosity model. It cannot be expected from a linear eddy viscosity model to predict the whole spectrum of anisotropy, since the turbulence anisotropy equals the strain rate anisotropy through the linear Boussinesq approximation. Nevertheless, these contours highlight the differences between the methods, in terms of turbulence character, while showing the spatial distribution. In summary, Fig. 5.5 demonstrates the complexity of the LPT flow and indicates that nearly all possible states of turbulence occur in different regions of the mid-span flow. The differences between the methods, in predicting the turbulence anisotropy, are not limited to the boundary layer region.

5.2.2 Entropy analysis

As already stated in Sec. 2.7, there is a general agreement that the entropy loss coefficient is one of the most accurate parameters for evaluating engine component performance. In this section, an assessment will be conducted of the flow losses in terms of entropy generation for the MTU-T161 test case at the mid-span. For that purpose, the analysis introduced in Sec. 2.7 will be utilized for both LES and RANS methods.

Figure 5.6 shows the normalized overall entropy generation rate predicted by LES and RANS. The term G_S , as already introduced in (2.103), consists of the following terms, mean viscous dissipation (V_M), turbulence (Reynolds stress) production (P_R) and turbulence advection (A_{TKE}). From the LES result, shown in the left column of Fig. 5.6, it can be concluded that more than 90 % of the overall entropy is generated in the separated shear layer over the separation bubble (grey dashed line). Additionally, the flow near the trailing edge area is a significant contributor to the overall entropy generation. This region is mostly affected by the small recirculation region near the trailing edge and the pressure side flow. The maximum of entropy generation occurs along the maximum height of the separation bubble (highlighted in Fig. 5.6).

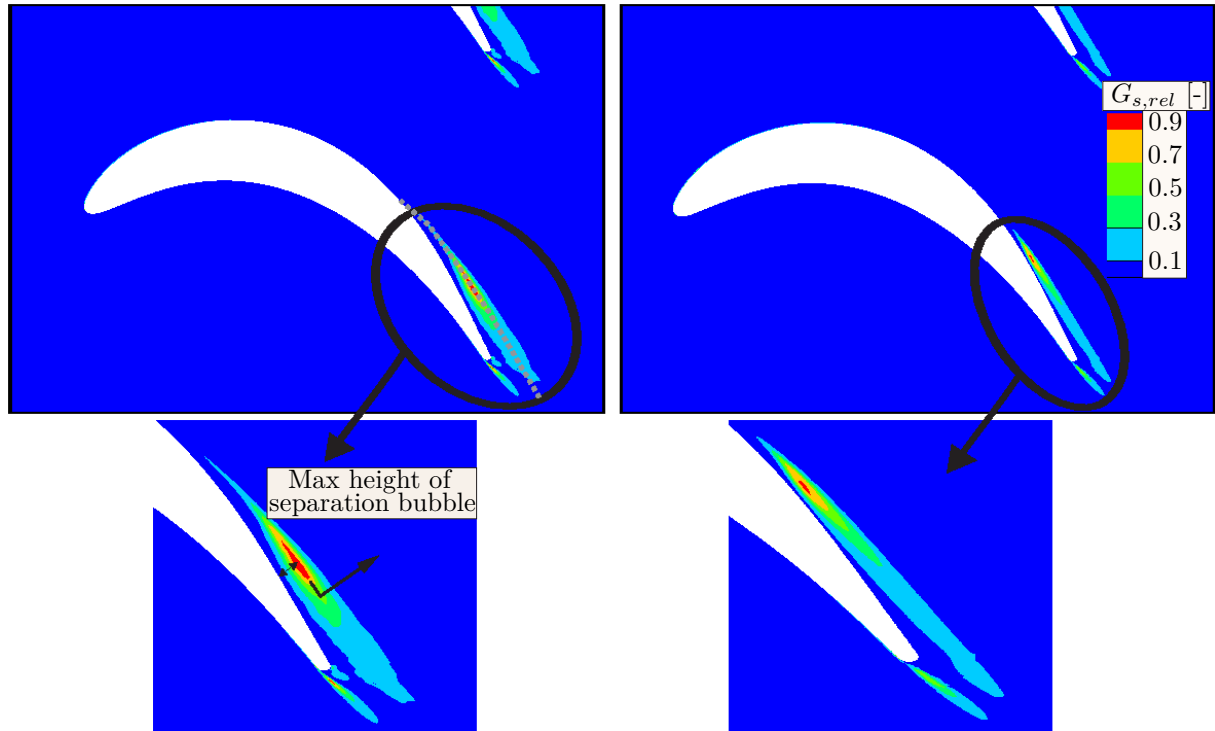


Figure 5.6: Normalized overall entropy generation rate cf. (2.103) for LES (left) and RANS (right) at mid-span of MTU-T161

By comparing these results with the results shown in Fig. 5.2 it can be concluded that there is a very close relation between the production of TKE and overall entropy generation. This hypothesis will be investigated in the next figures. Additionally, the vortex shedding of the separated shear layer (over the separation bubble) is one main cause for the excessive entropy generation. Some similar behaviors regarding the entropy generation can be found in the RANS result. But there are major differences regarding the shape and the region

of the high entropy generation rate. Both LES and RANS results show that most of the entropy is generated over the separation bubble and near the trailing edge. Nevertheless, the RANS result predicts these high entropy generation regions, to be at an earlier axial position and closer to the wall (over the suction side), than the LES result. Additionally, the entropy generation region is slightly thinner in the RANS results. By comparing the entropy generation contours with the TKE contours shown before, the RANS results (as in LES results) confirm the close relation between the TKE production and entropy generation.

The root-cause of the differences between the LES and RANS results, regarding the entropy generation, can be traced back to the different prediction of the separation bubble and the corresponding different turbulence behavior in this region. As already stated in Eq. 2.103, the overall entropy generation can be divided into three major contributors. In the next figures, the most important terms, namely the entropy generation related to turbulence production and turbulence advection, will be examined. Figure 5.7 shows the entropy generation related to turbulence production (P_R), cf. (2.103). The values are normalized by the maximum values of both LES and RANS individually. Figure 5.7 confirms the previous hypothesis that the overall entropy generation rate is mainly related to the turbulence production, created in the separated shear layer over the separation bubble. The results shown in Fig. 5.7 are very similar to Fig. 5.6 and Fig. 5.2.

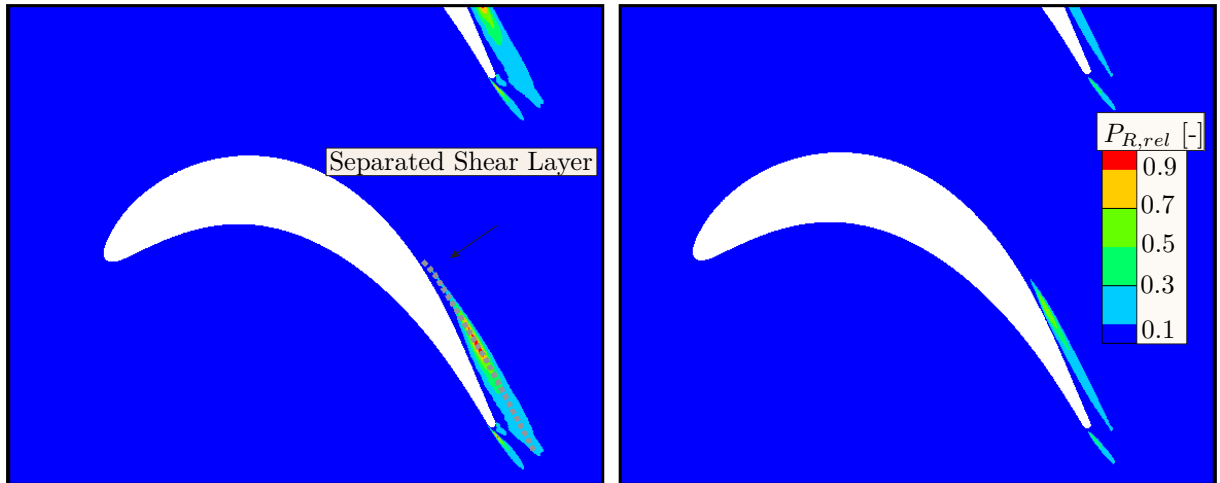


Figure 5.7: Normalized entropy generation rate, related to turbulence production, cf. (2.103) for LES (left) and RANS (right) at mid-span of MTU-T161

The RANS result is similar to LES, with major differences regarding the shape and positions of regions with high P_R values. These results show the importance of the correct prediction of the separation bubbles and related phenomena (transition to turbulence and reattachment) when using low order models such as RANS turbulence models. As shown earlier, cf. Fig. 5.3, the differences in TKE prediction between RANS and LES, start from leading edge, but is more pronounced in the separated region. The separated shear layer has the greatest influence on flow loss generation and turbulence anisotropy. Both methods predict the near trailing edge area, to be the second important region regarding the turbulence related entropy generation.

Figure 5.8 illustrates the combined role of the TKE and velocity gradient in the generation of entropy. The contours of A_{TKE} , complement the turbulence production contours. Most of the turbulence advection is generated in the layers slightly further away from the wall above the separation bubble (highlighted in the Fig. 5.8). In case of turbulence advection, the highest A_{TKE} values are generated near the trailing edge. RANS result, shows qualitative similar results as LES, with differences in shape and positions of the highest A_{TKE} values, as previously seen in P_R and G_s contours. The entropy generation contours

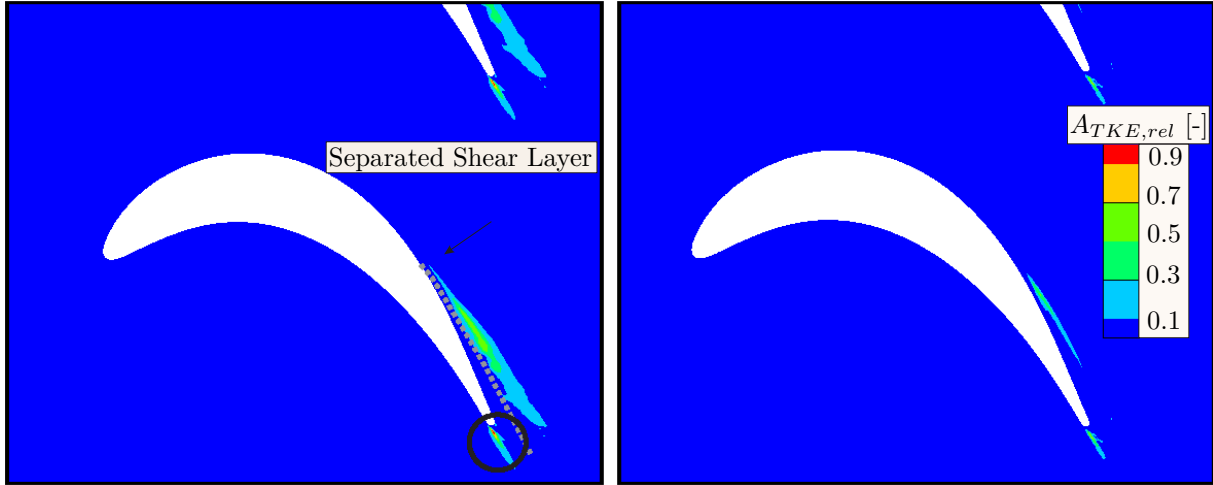


Figure 5.8: Normalized entropy generation rate, related to turbulence advection, cf. (2.103) for LES (left) and RANS (right) at mid-span of MTU-T161

revealed that the separated shear layer is the biggest contributor to entropy generation on the suction side, with P_R term playing the major role. The near trailing area, is the second contributor, with A_{TKE} term, mostly contributing to the entropy generation. On the other hand, the similarities and discrepancies between LES and RANS methods regarding flow loss prediction have been shown. RANS shows quantitatively similar results to LES, but the entropy generation levels are considerably lower and the shapes of the contours differ from the LES result, which has been shown to be mostly related to different prediction of separation bubble by each method.

Figure 5.9 shows the contribution of the major entropy generation terms in the mid-span wake plane, at $x/x_{lax} = 140\%$, for LES and RANS. Similarly, in the wake area (as in the passage flow), the turbulence production is mainly contributing to the entropy generation. The contribution of the turbulence production is followed by turbulence advection and mean viscous dissipation. In the wake area the contribution of mean viscous dissipation is zero, because of the free shear layer flow and non-existence of solid walls in this area.

The order of importance of the contributing terms is similar for both LES and RANS methods. But LES predicts higher values for turbulence production and advection than RANS. P_R values predicted by LES are up to 3 times higher than RANS. A_{TKE} values predicted by LES are even up to 6.5 times higher than RANS. The underprediction of the entropy generation levels in the wake area via RANS method, is consistent with the passage flow.

From the previous results it can be concluded that the main difference between LES and RANS results, in terms of entropy generation or flow loss prediction, originates from the

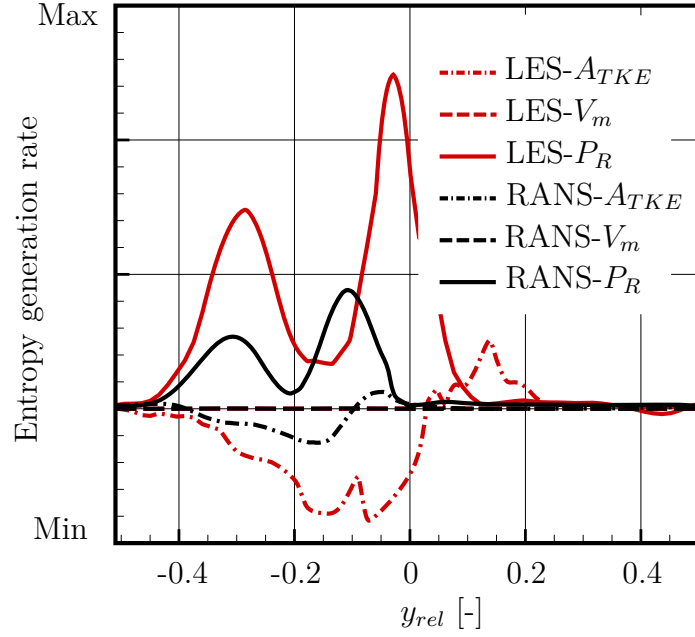


Figure 5.9: Entropy generation rate J/Ks (different terms) in wake region ($x/x_{lax} = 140\%$) at mid-span of MTU-T161 by LES (red) and RANS (black)

difference in modeling or resolving the turbulence production. This is, of course, the main task of turbulence modeling and shows the deficiencies of the turbulence model used in this work. At the same time, these results help us to understand the extent of the differences in terms of modeling the turbulence in the corresponding regions and its consequence for flow loss prediction. The findings are in accordance with the reduced total pressure loss in the wake region, as predicted by RANS, as outlined in Sec. 5.1.3.

5.2.3 Spectral analysis

In the current section, the spectral behavior of the suction side flow of the MTU-T161 cascade will be investigated. For this purpose, the time series data of the LES, stored in probes at different boundary layer cuts over the suction side, are used. The focus of this study is the suction side flow, as this region has been demonstrated to be of greater significance with regard to boundary layer dynamics and flow losses. The power spectral density (PSD) is calculated by Welch's method from the scipy module (Virtanen et al. (2020)). The original signal included 300 samples per convective time. A Hann window has been chosen with a window length of 10 convective times.

The resulting PSD of the streamwise velocity (velocity along the suction side surface) is shown for selected axial cuts on the suction side as a contour in Fig. 5.10. The x-axis represents the frequency multiplied by convective time and the normalized distance from the wall is shown on the y-axis. Additionally, the axial cuts, used for the PSD analysis are shown in Fig. 5.10 for orientation. It should be noted that the scales of the colorbars have been changed for each axial cut to make the details of the PSD contours more visible.

The first boundary layer cut is located in the laminar (pre-transitional) part of the boundary layer. At this location ($x/x_{lax} = 0.69$) there are two peaks below $ft_c < 1$ occurring

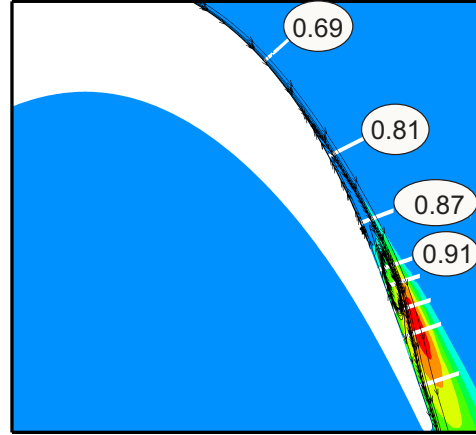
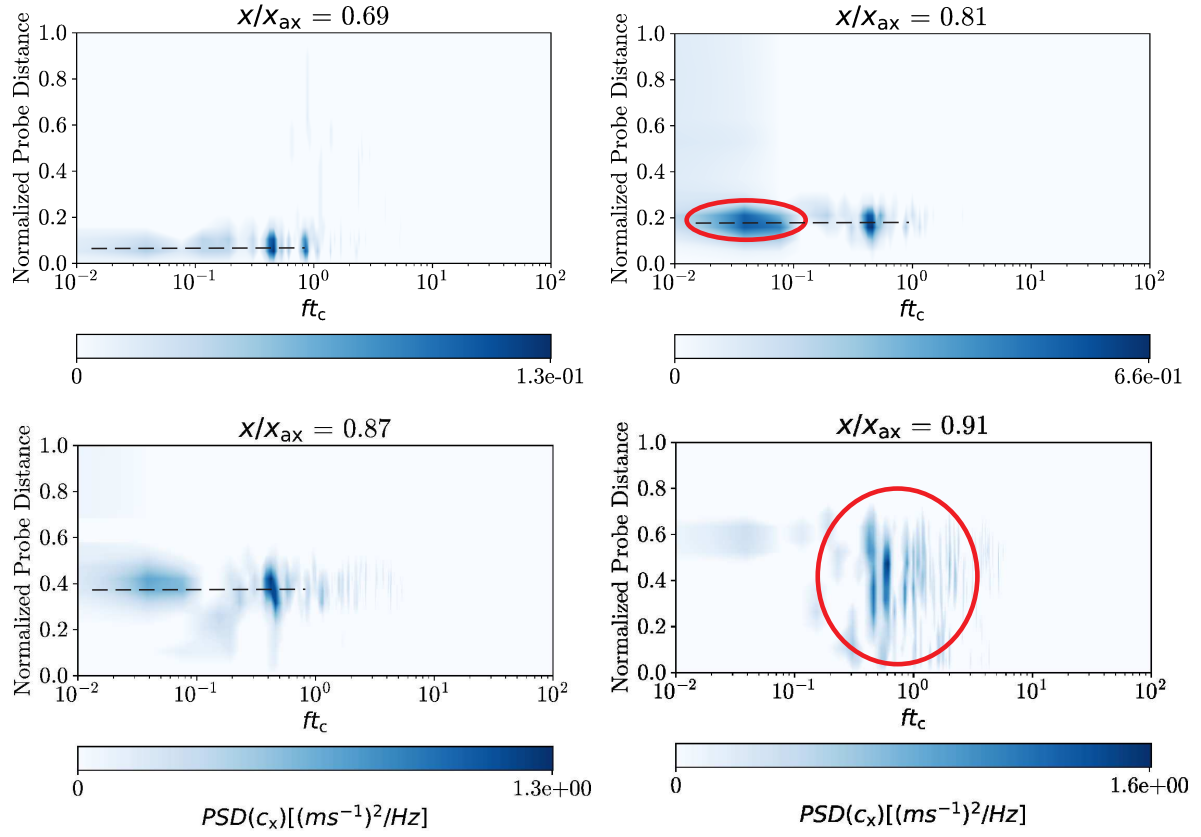


Figure 5.10: PSD of streamwise velocity for different boundary layer cuts on the suction surface along probes normal to the solid wall at the first half of the separation bubble. Zoomed blade to blade view and the positions of the corresponding boundary layer cuts with TKE contour. The PSD colorbars are scaled individually.

near the wall. The black dashed line highlights the wall normal position of the highest fluctuations. The cause of this amplification can be traced back to high shear rate at

this position. This finding is in line with the anisotropy behavior at this position, since high levels of shear in a flow lead to increased anisotropy (cf. Sec. 5.2.1). At the second cut, which is placed at the beginning of the separation bubble, there are fluctuation peaks at lower frequencies (than the first axial position). These peaks can be related to the flapping phenomenon. The flapping is an up-and-down motion of the separated shear layer (Spalart et al. (2000)). At this boundary layer cut, the frequency spectra in the front part of a separation bubble show sharp peaks at a very low frequency (flapping) and at a higher frequency (vortex shedding), therefore there are two fluctuating peaks. The spectral analysis of the first two boundary layer cuts represents and confirms the concept of shear sheltering introduced in Sec. 2.6.1.

Moving further downstream through the separation bubble at $x/x_{lax} = 0.87$, where the transition process starts, the peaks at low frequencies ($ft_c < 0.1$) fade out, and the dominant peak remains to be at $\approx ft_c < 0.5$. Additionally, fluctuations at higher frequencies ($ft_c > 1$) are observed at this boundary layer cut. At this axial point the Kelvin-Helmholtz instabilities play a major role in amplifying the fluctuations in higher frequencies. In the first three boundary layer cuts, the higher frequencies ($ft_c > 1$) are less pronounced, which suggests that the boundary layer is in its early stages and still developing. After reaching the middle of the separation bubble ($x/x_{lax} = 0.91$), no distinct frequency responsible for fluctuation peaks is observed. Instead, a distribution of fluctuation peaks over a wider frequency range is detected. At this axial point, the flow is fully developed and turbulent and this is the major cause for the different behavior of the fluctuations and instabilities.

Another interesting aspect is the wall-normal positions of the fluctuation peaks. While moving downstream, there is an elevation of the wall normal locations of the peaks, which is related to the separated shear layer moving away from the wall. At $x/x_{lax} = 0.91$, the fluctuations are visible along the shear layer as well as close to the wall. It should be noted that, the amplitude of the fluctuations continuously increases while moving further downstream across axial cuts.

In the second half of the separation bubble shown in Fig. 5.11, a similar behavior in the spectral analysis is observed as in the last axial cut in Fig. 5.10 (at $x/x_{lax} = 0.91$). At all axial positions in the second half of the separation bubble, the boundary layer is fully turbulent, characterized by broadband high-frequency fluctuations across broad wall normal positions. The amplitudes of the fluctuations increase continuously until reaching the axial cut at $x/x_{lax} = 0.94$. At this stage, turbulence production reaches its highest values, generating smaller turbulent structures and higher-frequency fluctuations. The high frequency fluctuations, are also related to the interaction of the shear layer and the free-stream turbulence. After reaching the maximum amplitudes of velocity fluctuations, the amplitudes decrease slightly in the last two axial cuts, which is related to the reattachment of the flow at this positions.

It is generally accepted that the presence of dominant peaks at different frequencies can be interpreted in the following manner

1. Fluctuation peaks at $ft_c < 1$ correspond to instabilities at low frequencies. These frequencies are lower than the inverse of the convective time scale ($1/t_c$) and signalize the presence of large-scale coherent structures. Such low-frequency peaks are associated with the separation of the flow and characterize the transition.

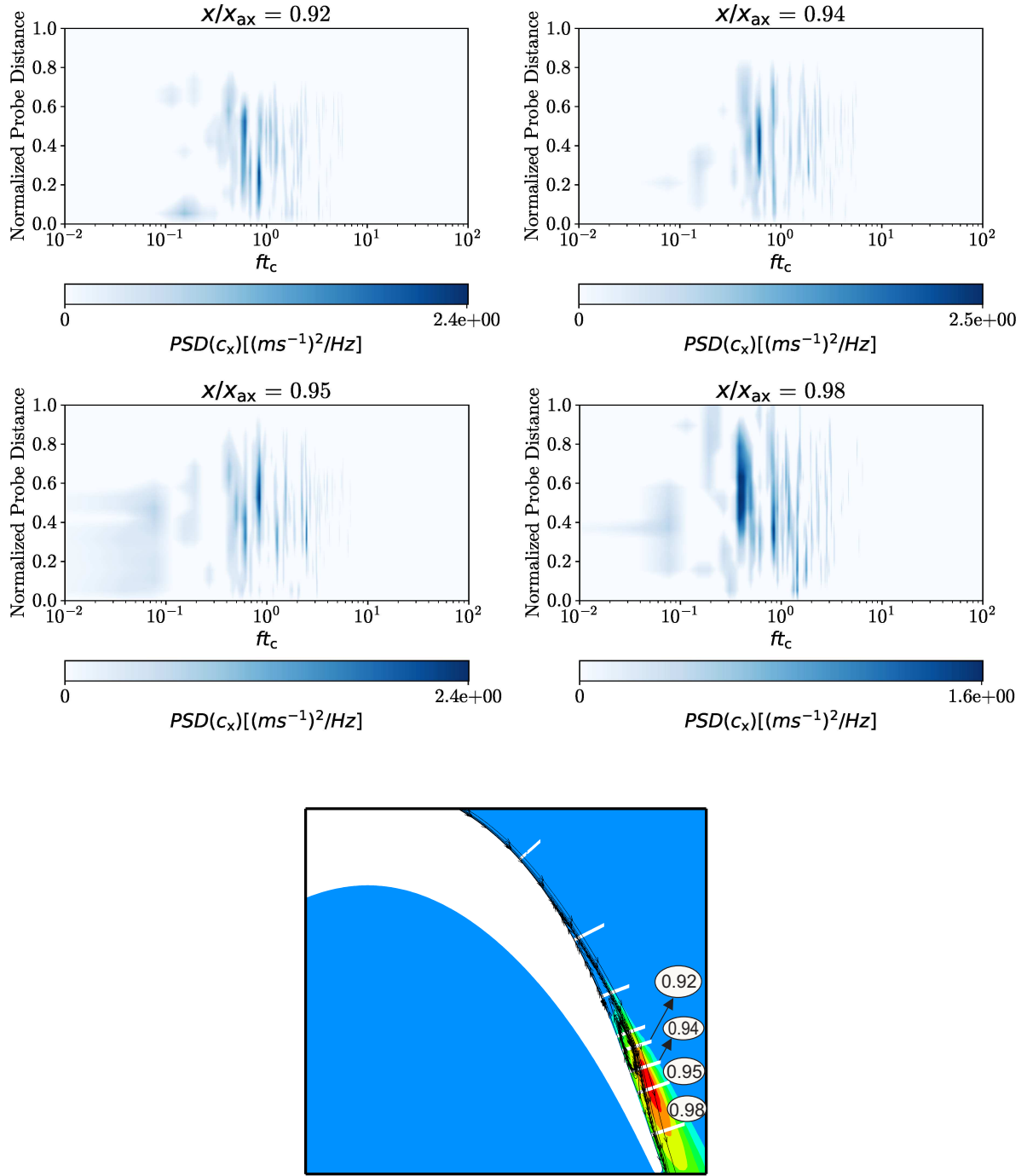


Figure 5.11: Power spectral density (PSD) of streamwise velocity for different boundary layer cuts on the suction surface along probes normal to the solid wall at the second half of the separation bubble. Zoomed blade to blade view and the positions of the corresponding boundary layer cuts with TKE contour. The PSD colorbars are scaled individually.

2. Fluctuation peaks at $ft_c > 1$ correspond to smaller-scale turbulent structures or higher-frequency instabilities in the corresponding flow regime. These fine-scale turbulent fluctuations arise, when the flow transition from laminar to turbulent, is further proceeded. In the case of flow separation, the higher frequencies correspond to vortex shedding across the separated shear layer and are related to the Kelvin-Helmholtz instability. These instabilities contribute to the rapid breakdown of the separated shear layer, leading to earlier transition in the LES results compared to RANS. The inability of RANS to resolve these structures can result in an overpredicted separation bubble length and an underestimation of turbulent diffusion into the wake.
3. The energy transfer from the low frequencies to higher frequencies reflects the classical turbulent energy cascade, where larger eddies (low frequencies) break into smaller turbulent structures while dissipating.

In the next section, the modal content of flow will be analyzed with the help of the SPOD method, to identify coherent structures contributing to the relevant frequencies.

5.2.4 Modal analysis

In order to analyze the spatial occurrence of the flow features, a spectral proper orthogonal decomposition (SPOD) is applied on a time series sampled at all nodes on the mid-span plane of MTU-T161. The SPOD is an empirical method for the extraction of coherent structures from the flow. The objective of SPOD is to identify an orthogonal basis in time and space for a given data set, whereby a small number of modes are capable of capturing a greater proportion of the total energy present in the data than any other orthogonal basis. Consequently, the modes are optimal for the description of the spatio-temporal coherence present in the data. The presented analysis closely follows the guidance and explanations provided by Schmidt et al. (2020). The SPOD analysis is performed using the Python package PYSPOD (Mengaldo et al. (2021)). For the SPOD analysis a total number of 20000 mid-span snapshots were collected over a period of 40 convective time with a frequency of 600,000 Hz. The data is collected far after the initial transient is passed. In this section, SPOD analysis, as explained in Sec. 2.8.1.2 and in App. A.2, is used to further investigate the MTU-T161 mid-span flow predicted by LES.

Figure 5.12 shows the spectrum of the eigenvalues for different mode numbers (mode indices). Each eigenvalue corresponds to the energy associated with a particular SPOD mode at a given frequency. Higher eigenvalues indicate dominant flow structures with significant energy. It is observed that the zeroth mode exhibits higher energy values than all the other modes. This behavior might be an identification that this mode captures the primary coherent structures in the flow (e.g., large-scale vortices or separation bubbles). This behavior is followed by mode 1, but the subsequent modes (with increasing number of mode indices), represent less energetic and less dominant structures, typically associated with smaller-scale phenomena or noise.

Figure 5.12 illustrates the spectral content of the modes, showing the corresponding frequencies responsible for the dominant peaks and the order of contribution of each mode, and provides the following insights. Most of the flow dynamics can be captured using a small number of modes, especially at low frequencies. The dominance of the first few

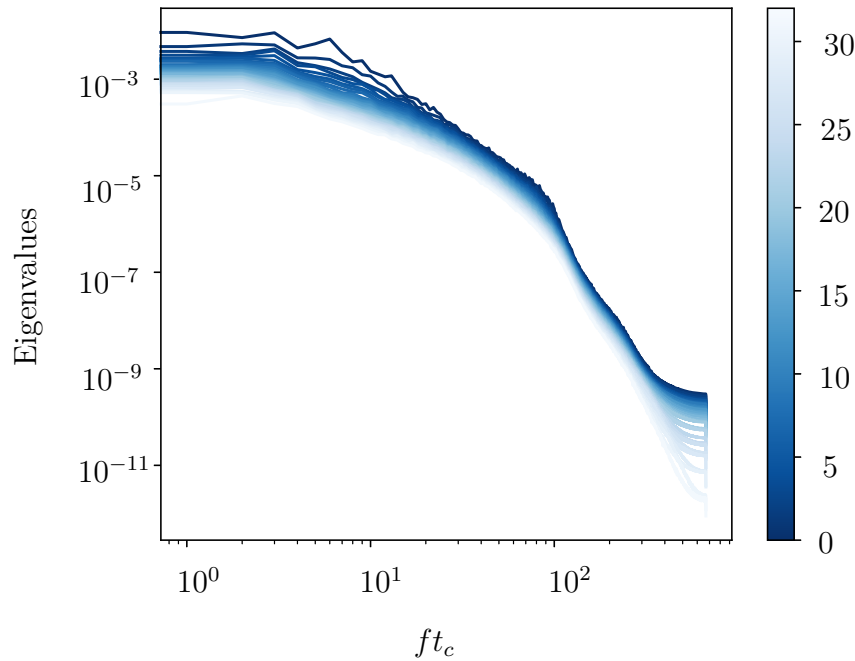


Figure 5.12: Eigenvalues of the SPOD modes over the frequency multiplied by convective time

modes at low frequencies reflects the presence of large, organized flow structures such as separation bubbles or coherent vortices. Figure 5.13 shows the amount of energy, each mode or eigenvalue, contributes to the overall energy. Eigenvalue energy captured is the sum of all the eigenvalues for all frequencies in a mode. The eigenvalue results are normalized by dividing each contribution by all the eigenvalues. This figure confirms the hypothesis before, that the first two modes (zero and one) are carrying most of the energy, but at the same time, a large number of modes to capture all the energy is needed. As a consequence, modes contributing less than 80 % to the overall energy were neglected.

In the next figures, the spatial distribution of the most important modes (based on normalized TKE), namely mode 0 and mode 1 for six distinct frequencies, will be examined. It should be noted that TKE was chosen for the SPOD analysis, since the focus in this work is primarily on TKE as an indicator of turbulence behavior.

Figure 5.14 shows the spatial distribution of the normalized TKE for mode 0 at six distinct frequencies. It is observed that the most dominant mode (mode 0) has the highest TKE content at $ft_c = 0.45$, which is associated with the separation bubble. At this frequency, there are two distinct regions, over the separation bubble and near wake, with high levels of TKE. These bulks of high TKE are convected further downstream by increasing the frequency.

At the second frequency ($ft_c = 0.9$), the highest level of TKE is shifted towards wake area. While increasing the frequencies, the structure and the spatial distribution of TKE, is not concentrated, but shows rather a vortex shedding character. At higher frequencies, the smaller-scale turbulent structures related to vortex shedding and fully turbulent flow, are more pronounced. The zeroth mode corresponds mainly to the kinetic energy associated with the mean flow. It captures the dominant coherent structures in the flow, such as separation bubbles, large vortices, and wake dynamics. Figure 5.15 shows the spatial dis-

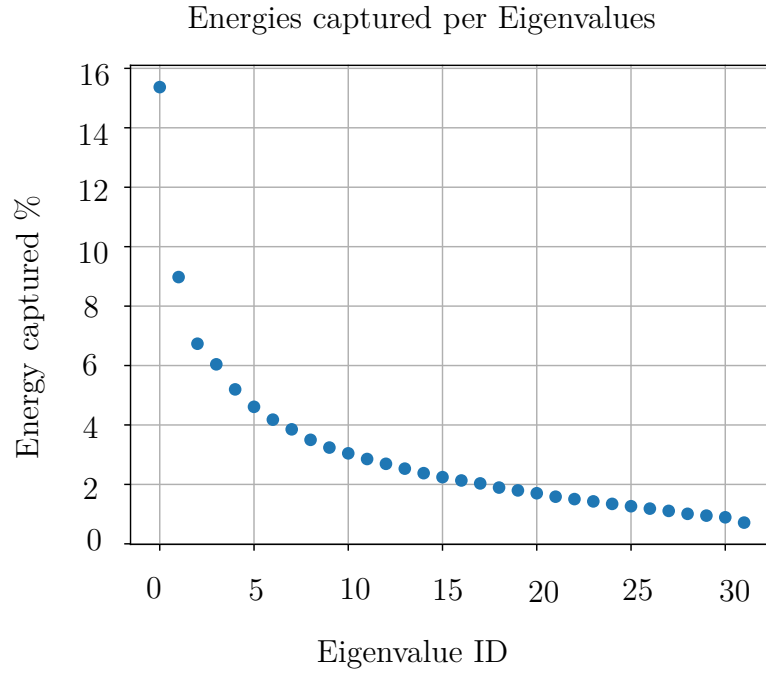


Figure 5.13: Normalized energy captured by each eigenvalue

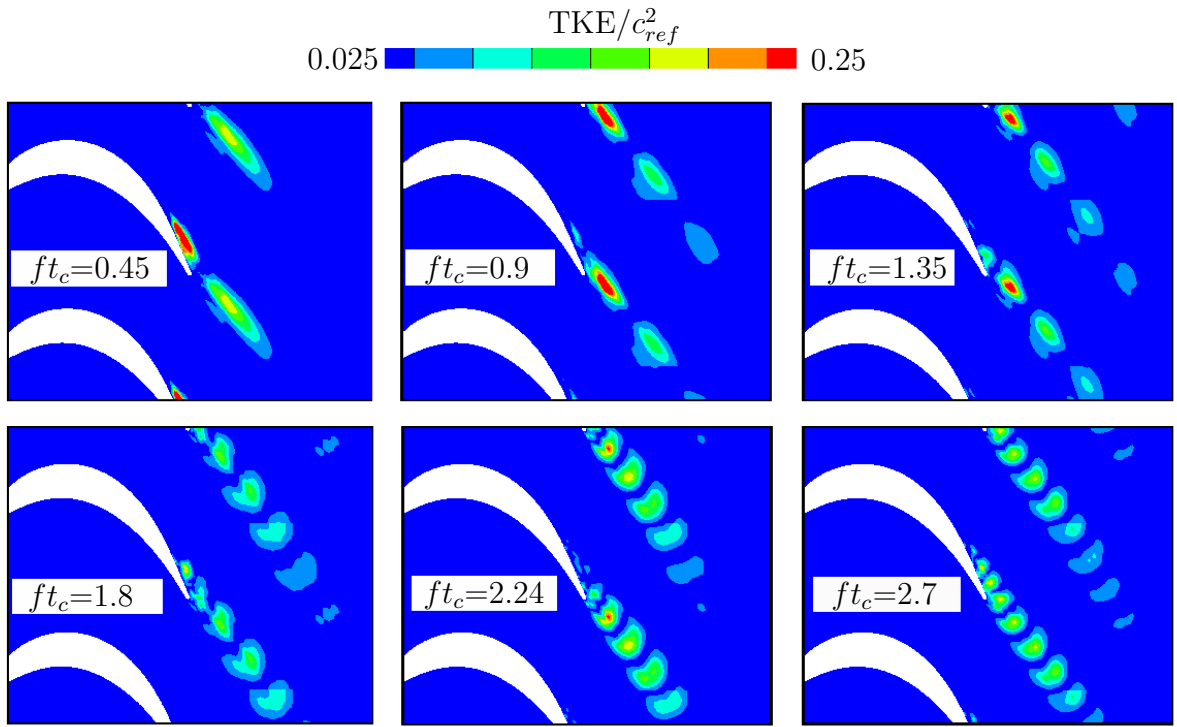


Figure 5.14: Mode 0 of the SPOD analysis of normalized TKE based on LES at mid-span plane of MTU-T161 at different frequencies

tribution of the normalized TKE for mode 1. Mode 1 is predominantly active in the region of the separation bubble. At $ft_c = 0.45$, the energy is concentrated further downstream compared to mode 0, indicating that structures are influenced by the dynamics of mode 0. At mode 1, the presence of secondary vortices or weaker flow oscillations is observed. The footprint of the separated shear layer over the separation bubble is observed. It is

evident that an increase in frequency results in a vortex shedding phenomenon within the flow, particularly in the reattachment region. It can be concluded that mode 1 captures less organized, higher-frequency turbulent features influenced by mode 0 shedding. The

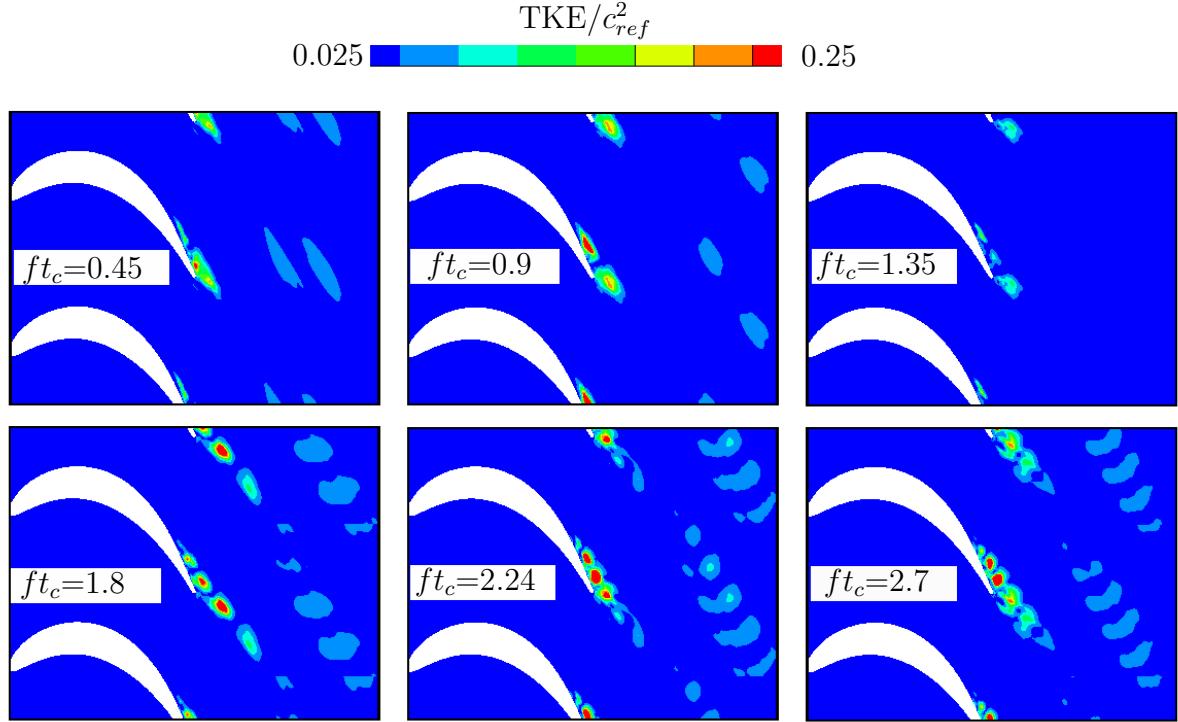


Figure 5.15: Mode 1 of the SPOD analysis of TKE based on LES at mid-span plane of MTU-T161 at different frequencies

results of other modes are not shown here, since the contribution of each mode decreases with increasing mode number. The results of SPOD show the different character of each mode, while each mode complementing the whole flow picture. Mode 0 shows the dominant coherent structures responsible for separation bubble and wake vortices. Mode 1 captures secondary, less energetic structures that complement mode 0, often representing wake instabilities or interactions between different scales. Additionally, it is shown how the SPOD analysis can capture different dynamics in the flow and relate the flow phenomena to each mode, frequency, and the corresponding spatial region.

5.2.5 Quadrant Analysis

In this section, the quadrant analysis, introduced in Sec. 2.8.2, is used to investigate the identification of the principal contributions to turbulence (based on Reynolds stresses). For the Reynolds stress quadrant analysis, the Reynolds shear stress ($c_x'c_y'$) is split into four quadrants depending on the sign of the streamwise c_x' and the wall-normal velocity fluctuation c_y' . The results presented in this section facilitate comprehension of the role of shear stress on turbulence dynamics. This analysis serves as a valuable complement to the TKE analysis conducted previously. The following figures depict the quadrant analysis for the Reynolds stress component ($c_x'c_y'$) for different boundary layer sections at the mid-span flow of MTU-T161 using the LES data. The boundary layer sections are

located before separation ($x/x_{lax} = 70\%$), after separation ($x/x_{lax} = 80\%$), in the middle of the separation bubble ($x/x_{lax} = 90\%$) and after reattachment ($x/x_{lax} = 98\%$). At each boundary layer section, five positions normal to the suction side surface are selected to plot the contribution of each quadrant to the Reynolds shear stress. Each figure represents the ratio of the total time the flow spends in each quadrant at different axial positions and normalized distances from the suction side of the turbine blade.

At the first boundary layer section, Q2 and Q4 are dominant throughout all points normal to the wall. The prevalence of Q2 and Q4 events decreases with distance from the wall. High values of Q2 highlight ejections that likely involve low-momentum fluid being pushed away from the boundary layer into the free-stream. High values of Q4 highlight sweep events that high-momentum fluid from the free-stream is brought towards the wall. After moving further away from the wall, the contribution of Q1 and Q3 become more prominent, while Q4 and Q2 are still the biggest contributor to the Reynolds shear stress. Q1 and Q3 have minimal representation, suggesting infrequent inward and outward movements of fluid towards the wall at this pre-separation point. Q1 includes outward motion of high-velocity fluid and Q3 represents the inward motion of low-velocity fluid.

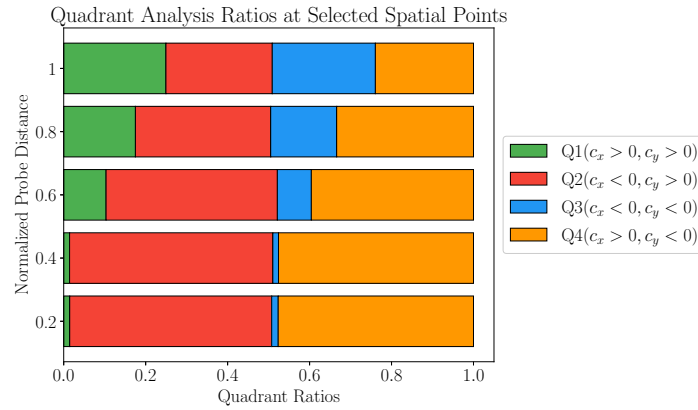


Figure 5.16: Quadrant analysis at mid-span of MTU-T161 based on LES at boundary layer section $x/x_{lax} = 70\%$

When moving to the second and third boundary layer cut at $x/x_{lax} = 80\%$ and $x/x_{lax} = 90\%$, there is a slight increase of Q1 and Q3 events compared to the pre-separation (pre-transitional) phase, particularly in the third axial position ($x/x_{lax} = 90\%$), reflecting enhanced turbulent mixing as the separation process begins. A slight increase of Q3 potentially indicates the inward transport as the separated flow starts to interact with the free-stream turbulence. Q2 is prominent across a range of wall-normal distances. These ejections indicate a significant presence of upward-moving turbulent bursts, which are characteristic of separated flow regions where low momentum fluid is detached from the wall. Q4 remains strong near the wall, especially in the third axial position ($x/x_{lax} = 90\%$), suggesting ongoing high-momentum sweep events that contribute to maintaining some level of momentum near the wall despite separation. Starting from the axial position $x/x_{lax} = 90\%$, the contribution of Q1 and Q3 becomes more significant, apart from the near wall regions.

At the last boundary layer section ($x/x_{lax} = 95\%$), where the flow is reattached and fully turbulent, the order of the quadrant ratios changes slightly compared to the previous axial

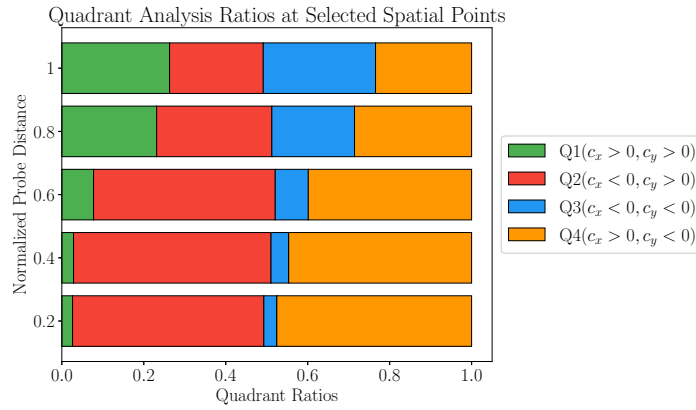


Figure 5.17: Quadrant analysis at mid-span of MTU-T161 based on LES at boundary layer section $x/x_{lax} = 80\%$

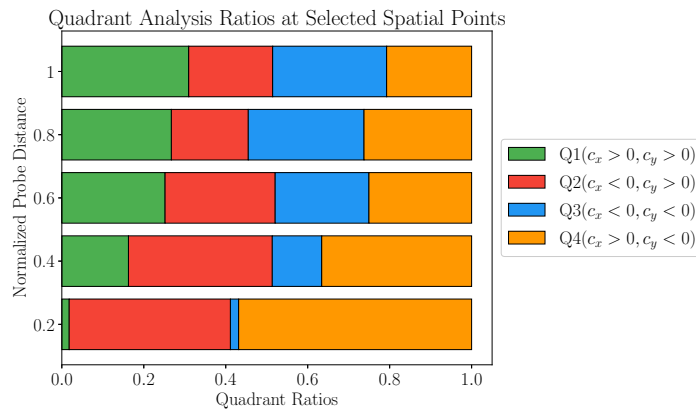


Figure 5.18: Quadrant analysis at mid-span of MTU-T161 based on LES at boundary layer section $x/x_{lax} = 90\%$

boundary layer cuts. Q1 and Q3 events are more prominent (than the other two quadrants) when moving away from the wall. On the other hand, Q2 and Q4 still dominate the region near the wall, highlighting recovering boundary layer, where ejections help in mixing and redistributing momentum and at the same time continued influence of sweeps in bringing high-momentum fluid towards the surface, helping in boundary layer stabilization and boundary layer thickening in post-reattachment region.

Quadrant analysis reveals a transition in the suction side flow of the low-pressure turbine blade, from attachment to separation and reattachment, along the streamwise direction. Notably, the dominant Q4 (sweep) events near the wall across all figures indicate a significant role of high-momentum fluid transport towards the surface, impacting the wall shear stress distribution. This behavior is mostly because of the presence of the free-stream turbulence. The Q2 (ejection) events, prevalent especially during separation and post-reattachment, illustrate the dynamic nature of the boundary layer, characterized by the ejection of low-momentum fluid. Q2 and Q4 events contribute positively to the TKE production. At the same time, the opposite signs of c_y' fluctuations imply that these events are individually responsible for energy flux towards and away from the wall.

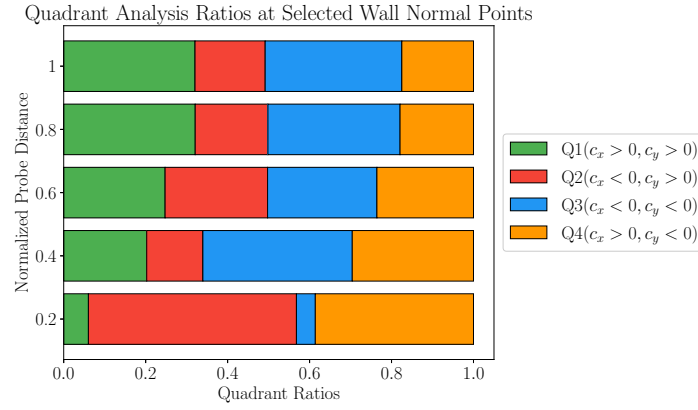


Figure 5.19: Quadrant analysis at mid-span of MTU-T161 based on LES at boundary layer section $x/x_{lax} = 95\%$

In the literature, it has been shown for canonical cases, that the flux from the outer region increases the adverse pressure gradient and changes the pressure distribution and skin-friction (Deshpande et al. (2024)). The presence of Q1 and Q3, though less prominent, provides a complete picture of the complex interactions occurring in the boundary layer, with outward and inward fluid motions contributing to the overall turbulent structure. It can be concluded that the turbulent ejections from the wall and sweeps are the dominant turbulent mechanism for the generation of turbulence-driven secondary flow. The TKE distribution identifies the regions of strongest turbulence activity. The quadrant analysis then reveals that in these regions, ejections and sweeps are the dominant contributors, consistent with strong shear-layer turbulence production.

5.3 Influence of the free-stream turbulence on the flow at mid-span

In the previous sections, the mid-span flow of MTU-T161 is analyzed with the help of LES results, regarding flow losses and turbulence behavior and these results are compared with best-practice RANS results. In the current section, the impact of varying free-stream turbulence levels on the mid-span flow characteristics of the MTU-T161 turbine blade is examined for the operating point $Re_{2,th} = 90,000$, $M_{2,th} = 0.6$. The focus is on critical design parameters, including the isentropic Mach number distribution along the blade surface and the total pressure loss in the wake. To achieve this, three LES simulations were conducted with different inlet free-stream turbulence intensities, denoted as LES-Clean, LES-4% and LES-8%. These cases were conducted without varying the integral turbulent length scale.

It is well-established in the literature that turbulence intensity and turbulent length scale both play crucial roles in influencing turbulence behavior and boundary layer development. In this context, the inflow turbulence generation method in TRACE (cf. 4.2) uses a correlation between the prescribed turbulent length scale and the computational domain's dimensions. However, this approach imposes constraints, particularly in three-dimensional configurations where the pitch dimension becomes the limiting spatial factor.

For this study, the maximum turbulent length scale allowed by the 3D configuration was employed. Simulating larger turbulent length scales would have required expanding the domain beyond a single pitch, which was computationally impractical. Consequently, the analysis is limited to a single passage, highlighting the trade-offs between computational feasibility and capturing more complex flow physics. This constraint aligns with findings from prior studies, which emphasize the importance of achieving a balance between domain size, grid resolution, and computational cost when investigating turbulence effects in turbomachinery.

Figure 5.20 presents the isentropic Mach number and skin-friction distribution along the blade profile for all three cases. The LES-Clean case exhibits notable deviations compared to the other two cases. Most prominently, the flow in the LES-Clean case experiences massive separation on the suction side without subsequent reattachment. The absence of free-stream turbulence prevents the flow from reattaching, and results in an extensive separation region, which is evident in the isentropic Mach number distribution over the suction side and at the trailing edge (LES-Clean case). The early separation on the suction side, occurring approximately 15 % closer to the leading edge than to the other cases, is clearly depicted in the skin-friction development (Fig. 5.20).

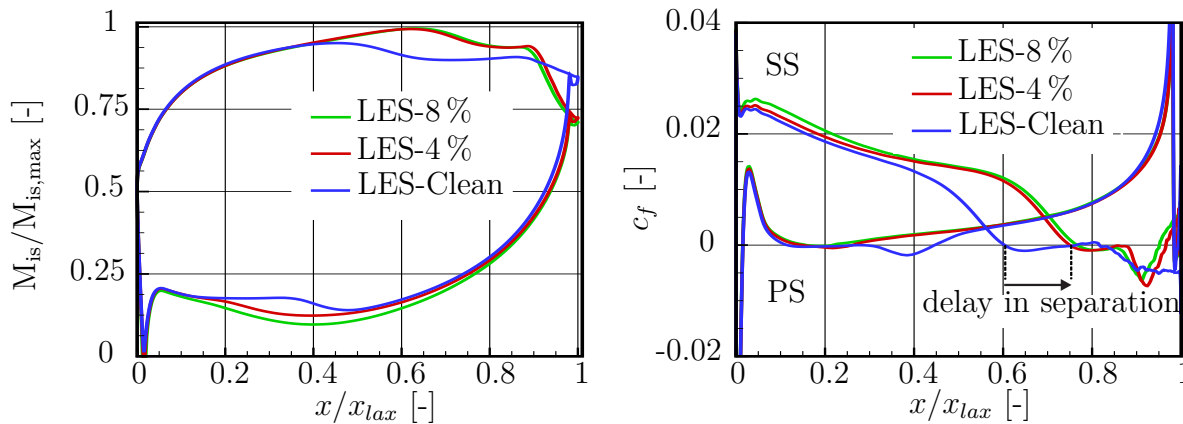


Figure 5.20: Isentropic Mach number and skin-friction coefficient at mid-span of MTU-T161 by LES for three free-stream turbulence levels

The pressure side flow is also significantly influenced by the absence of free-stream turbulence. A larger separation bubble forms on the pressure surface in the LES-Clean case, with its axial extension being 2.8 times greater than the other two cases. This highlights the critical role of free-stream turbulence in stabilizing boundary layer behavior on both suction and pressure sides. The results of the LES-4 % case were already discussed in the previous sections (cf. 5.1.1), therefore the focus will be on the LES-8 % results and the effects of increased free-stream turbulence intensity (doubling the FSTI) on the flow.

Doubling the free-stream turbulence intensity, leads to slight changes (compared to LES-4 % case) in the flow on both the suction and pressure sides. On the suction side, the separation occurs later, and reattachment happens earlier, reducing the size of the separation bubble compared to the other two cases. Specifically, the separation bubble length decreases by a factor of 1.2, compared to LES-Clean. On the pressure side, the separation bubble becomes substantially smaller with increased free-stream turbulence (compared to

LES-Clean). The reduction in separation bubble size can be attributed to enhanced entrainment mechanisms driven by the free-stream turbulence. The free-stream turbulence increases the entrainment rate, which balances the fluid recirculated into the cavity region by the reattaching shear layer and the entrainment into the shear layer near the separation point. The separation bubble is maintained by two opposing processes. Recirculation, which is the fluid within the bubble circulates and sustains the bubble's structure. Entrainment, which is the fluid from outside the bubble is pulled in, disrupting the bubble's stability and encouraging it to shrink or collapse. When free-stream turbulence is present, it makes the entrainment process more effective. This means the bubble can't sustain its size because more fluid is being pulled into the shear layer and mixed away. As a result, the bubble becomes smaller.

In a more global sense, the blade loading increases with increasing the free-stream turbulence intensity, underscoring its influence on both boundary layer dynamics and aerodynamic performance (cf. Fig. 5.20). These findings align with existing literature (Wang et al. (2021), Jeong et al. (2024)), which emphasizes the role of turbulence in mitigating separation and improving flow reattachment. Figure 5.21 depicts the evolution of streamwise velocity along the suction side of the blade for all three cases, offering a clear visualization of the differing disturbance mechanisms on the suction surface. In the LES-Clean case, no disturbances are observed until the flow reaches the separation point, approximately halfway along the suction side. Conversely, the cases with free-stream turbulence exhibit streamwise fluctuations beginning at the leading edge, with the LES-8 % case, as expected, showing the highest levels of fluctuations.

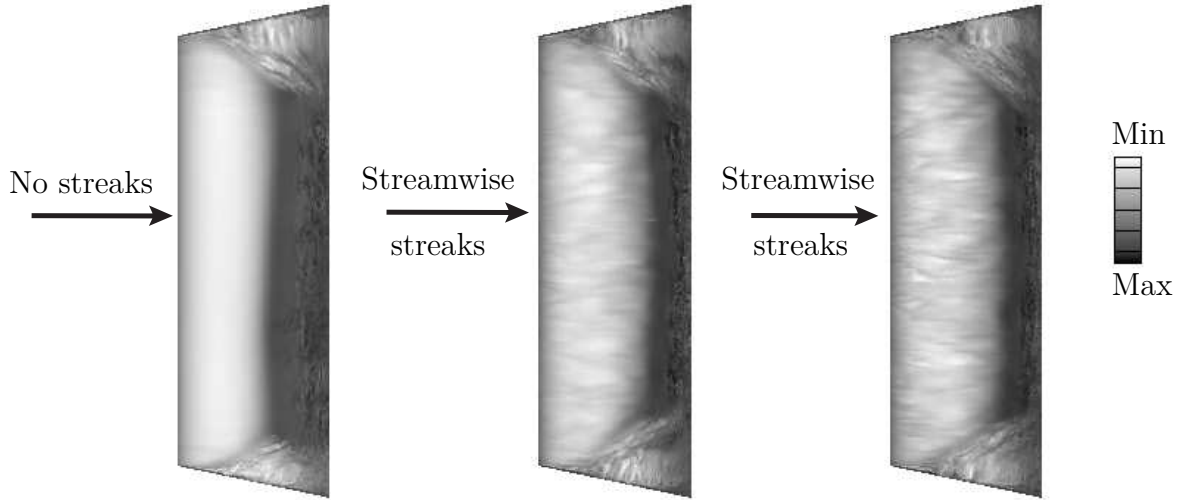


Figure 5.21: Contours of the instantaneous streamwise velocity field to visualize the formation of Klebanoff-modes on the suction side at mid-span of MTU-T161. Influence of the variation of free-stream turbulence levels ($TI = 0\%$ to $TI = 8\%$) on the reattachment. Left: LES-Clean, Middle: LES-4 %, Right: LES-8 %

Figure 5.21 also highlights the formation of Klebanoff modes under varying free-stream turbulence intensities. These modes, first described by Klebanoff et al. (1962), are a critical aspect of transient growth mechanisms influenced by free-stream turbulence. They manifest as elongated structures aligned with the flow direction and exhibit rotational behavior around the flow vector. Typically observed at low turbulence intensities such

as $TI \approx 0.5\%$, Klebanoff modes are characterized by low-frequency fluctuations due to their large wavelength. In Sec. 5.2.3, the presence of low frequency disturbances for the LES-4 % case, was shown using a spectral analysis. Klebanoff modes play a pivotal role in boundary layer transition by stimulating Kelvin-Helmholtz (KH) instability, which forms small vortex loops around the modes. These small-scale vortices interact with the KH instability, disrupting its spanwise roll-up and accelerating the transition to turbulence.

At higher free-stream turbulence levels, $TI \approx 3\%$, the exponential growth of boundary layer disturbances becomes dominant, as noted by Brandt et al. (2004). Depending on the turbulence intensity, Klebanoff modes can either grow linearly, leading to the formation of λ -structures, or undergo exponential growth, directly forming turbulence spots (McAuliffe et al. (2010), Balzer et al. (2016)). In case of free-stream turbulence, the streaks penetrate through the boundary layer causing the streamwise streaks, which is referred to in the literature as shear shelter, and was previously confirmed by spectral analysis (cf. Fig. 5.10).

Figure 5.22 illustrates the evolution of normalized maximum turbulent kinetic energy (TKE) along the suction side for all three cases, providing insight into the impact of free-stream turbulence on the TKE distribution. In the LES-Clean case, TKE remains

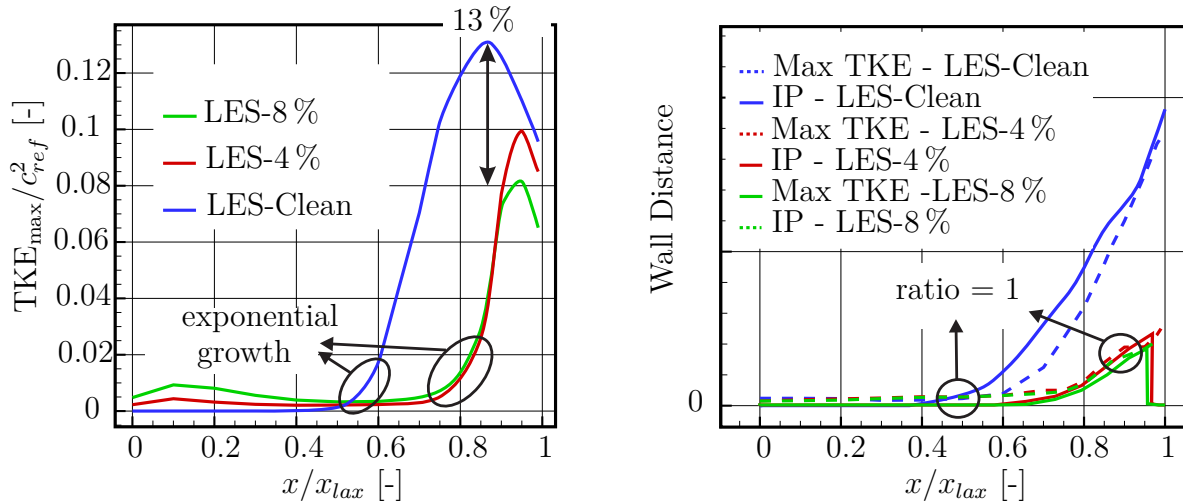


Figure 5.22: Evolution of normalized maximum TKE (left) and wall normal position of maximum TKE and inflection point (IP) (right) along the suction side flow at mid-span of MTU-T161 by LES for three free-stream turbulence levels

at low levels until approximately $x/x_{lax} = 50\%$ (separation point). Beyond this point, following flow separation, the normalized TKE increases exponentially, which is a sign of inflectional-instability mechanism and predominance of Kelvin-Helmholtz (KH) vortices, followed by a linear growth at around $x/x_{lax} = 60\%$. The TKE value reaches its peak values near $x/x_{lax} = 85\%$. Additionally, the LES-Clean case shows higher TKE_{max} values (up to 13 % higher), than the other two cases, at an earlier axial position (8 % shift towards leading edge). The results of the LES-Clean case show that the KH instability is mainly promoting the transition in the case without free-stream turbulence.

In contrast, the LES-4 % and LES-8 % cases exhibit similar trends in TKE evolution, with only minor variations. The LES-8 % case consistently shows higher TKE values compared

to the LES-4 % case. Interestingly, despite higher TKE levels during the buildup phase ($0 < x/x_{lax} < 0.5$), the LES-8 % case reaches a lower TKE peak than the other two cases after the exponential increase begins. Both cases demonstrate non-zero TKE values ahead of separation, which is another sign of the Klebanoff modes (streamwise streaks) in these regions. This behavior is also shown qualitatively in Fig. 5.21. As in the LES-Clean case, both free-stream cases show exponential evolution of instabilities, followed by linear growth. But the free-stream cases, encounter the KH instability further downstream, namely over the separation bubble. The lower TKE-peak for the LES-8 % case can be traced back to the following phenomena

1. The separation bubble on the suction side (compared to the other cases) is smaller, and the associated large-scale recirculation is weaker. Since large-scale separation regions contribute significantly to TKE production, their suppression reduces the peak TKE.
2. High free-stream turbulence introduces a broader spectrum of disturbances. This energy is distributed across a wide range of scales rather than concentrated in large-scale structures, which dominate TKE in cases with lower free-stream turbulence. Consequently, the TKE-peak appears lower, even though the overall turbulence intensity is higher.
3. Increased free-stream turbulence enhances turbulent mixing within the boundary layer. This efficient mixing redistributes energy across the boundary layer, preventing pronounced peaks of TKE.

Additionally, Fig. 5.22 displays the wall-normal position of maximum TKE and the inflection point (IP) for all three cases, along the suction surface. The ratio of Inflection point to wall normal position of max TKE (when the two lines cross) can be interpreted, according to Diwan et al. (2009), as following

1. If the ratio (IP / wall normal position of TKE_{max}) is $\ll 1$, wall mode is dominant
2. If the ratio (IP / wall normal position of TKE_{max}) is ≈ 1 , the inflectional mode (Kelvin-Helmholtz) controls the disturbance dynamics

Figure 5.22 (right side) shows that in the LES-Clean case, the two lines cross each other at around $x/x_{lax} = 50\%$. This fact suggests that the dominant transition mechanism is associated with the inflectional mode in the early stage of separation. The other two cases, demonstrate the ratio of unity around the $x/x_{lax} = 90\%$, which is related to the KH instability over the separation bubble. The results in Fig. 5.22 confirm the previous findings.

The influence of the massive separation and the open separation bubble on the total pressure loss in the wake area (at $x/x_{lax} = 1.4$), is depicted in Fig. 5.23. In the LES-Clean case, the wake flow loss exhibits significantly higher peak values and wider flow loss curve compared to the other cases. As the free-stream turbulence intensity increases, the wake flow loss continuously decreases. The integral flow loss for the LES-Clean case is up to 2.8 times higher than the other two cases. Similarly, the integral flow loss for LES-4 % is 1.13 times higher than LES-8 %.

To better understand the role of turbulence in shaping the separation bubbles, subsequent figures analyze the turbulent kinetic energy (TKE) budget terms, offering valuable insights

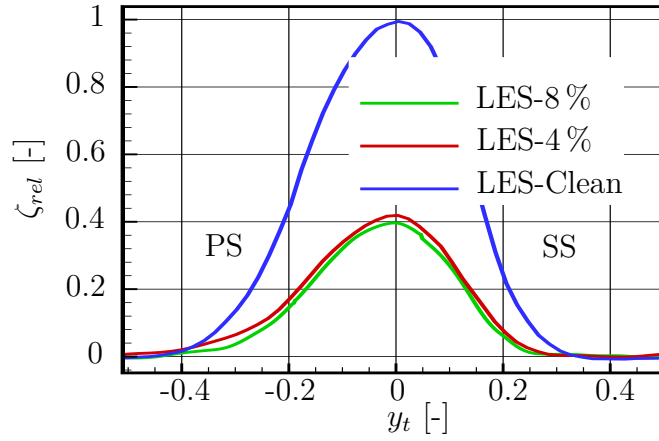


Figure 5.23: Normalized total pressure loss in wake region ($x/x_{lax} = 140\%$) at mid-span of MTU-T161 by LES for three free-stream turbulence levels

into the interplay of various physical phenomena. First, contours of TKE and shear stress are presented to provide an overview of turbulence distribution. The analysis then focuses on specific terms contributing to the TKE budget, including TKE production (P), TKE dissipation (ϵ), TKE convection and TKE diffusion (turbulent D_j^u and pressure diffusion D_j^p). This detailed breakdown helps elucidate the mechanisms influencing the flow and its flow losses.

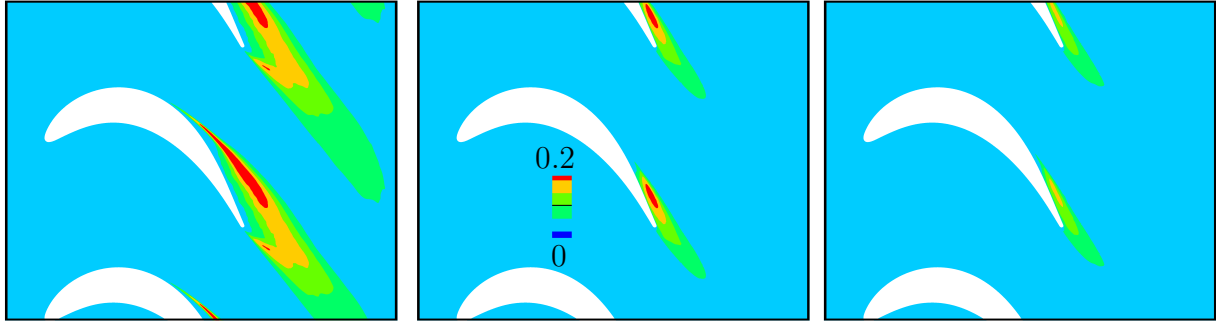


Figure 5.24: Normalized TKE contours at mid-span of MTU-T161 by LES for three free-stream turbulence levels. Left: LES-Clean, Middle: LES-4 %, Right: LES-8 %

Figure 5.24 illustrates the spatial distribution of turbulent kinetic energy (TKE) for all three cases, highlighting distinct patterns influenced by free-stream turbulence levels. The highest TKE values are primarily observed within the separated shear layer for the LES-Clean case and additionally in the reattachment region for the cases with free-stream turbulence. As the free-stream turbulence intensity increases, regions of elevated TKE shrink and shift further downstream, concentrating towards the latter half of the separation bubble and the reattachment zone. In the LES-8 % case, the peak TKE levels are notably lower than those observed in the other cases, which is consistent with the findings in Fig. 5.22. This reduction emphasizes the role of higher free-stream turbulence in stabilizing the flow and suppressing large-scale instabilities within the boundary layer. The contours highlight the significant impact of the massive separation (LES-Clean) on TKE generation, as the unsteady separated shear layer contributes extensively to turbulence production.

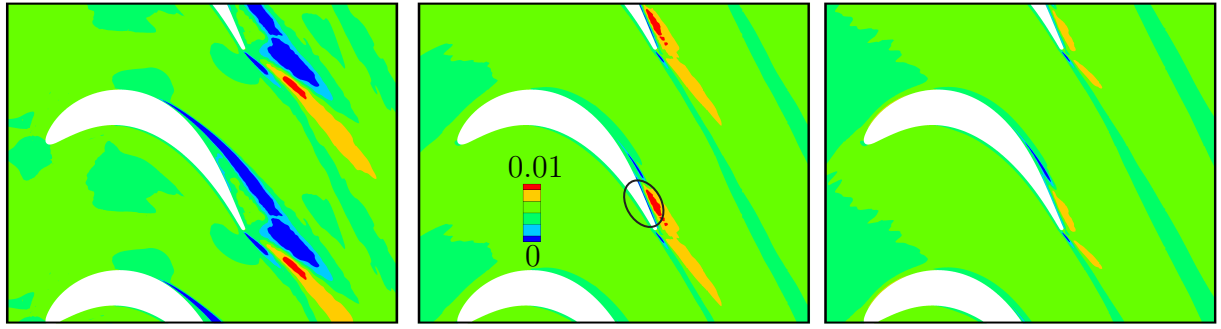


Figure 5.25: Normalized shear stress contours at mid-span of MTU-T161 by LES for three free-stream turbulence levels. Left: LES-Clean, Middle: LES-4 %, Right: LES-8 %

Similarly, the shear stress contours (Fig. 5.25) reveal high values along the separated shear layer for the LES-Clean case, with the maximum negative shear stress concentrated in the unsteady, separated region. The blue regions indicate regions with flow separation or regions with adverse pressure gradients. The red regions indicate turbulent, thin boundary layers with strong momentum gradients and higher shear stresses. For the cases with free-stream turbulence, the highest shear stress values occur in the second half of the separation bubble, the reattached and fully turbulent regions. Consistent with the TKE results, the LES-8 % case exhibits lower maximum shear stresses compared to the other two cases. This outcome underscores the stabilizing influence of increased free-stream turbulence, which promotes earlier transition and reduces the intensity of large-scale flow structures. The thin blue region near the trailing edge, on the suction side (next to the wall) of LES-4 %, highlights the localized separation. The separation is due to the adverse pressure gradient weakening the boundary layer, and not sufficient free-stream turbulence intensity.

Figure 5.26 illustrates the ratio of turbulent kinetic energy (TKE) production to dissipation, providing insight into the TKE imbalance at the mid-span of the low pressure turbine cascade flow. Figure 5.26 reveals that in the separated shear layer, turbulence dissipation plays a minor role, leading to a significant imbalance with the production dominating strongly. The ratio $\frac{P}{\epsilon}$ reaches values as high as 20, highlighting the dominance of TKE production in this region. This imbalance indicates the shear layer's role as a highly active zone of energy transfer, driven predominantly by production mechanisms. In Sec. 3.1.3 the important role of TKE imbalance e.g. non-equilibrium in cascade flow was shown.

Figure 5.27 depicts the sum of turbulent and pressure diffusion for all three cases. In the cases with free-stream turbulence, diffusion begins abruptly in the reattachment region, driven by the roll-up of Kelvin-Helmholtz vortices, rapid shear layer spreading, and subsequent turbulence breakdown. These processes underscore the critical role of diffusion in facilitating reattachment. Both free-stream cases are very similar in terms of turbulence diffusion behavior. For the LES-Clean case, elevated TKE diffusion values are observed along the separated shear layer and in the wake region. The absence of reattachment in the LES-Clean case contributes to these differences, as the non-reattaching separation bubble alters the energy redistribution.

Turbulence convection (Fig. 5.28) follows a similar pattern to diffusion. In the LES-Clean case, high convection values are observed along the separated shear layer, while the cases

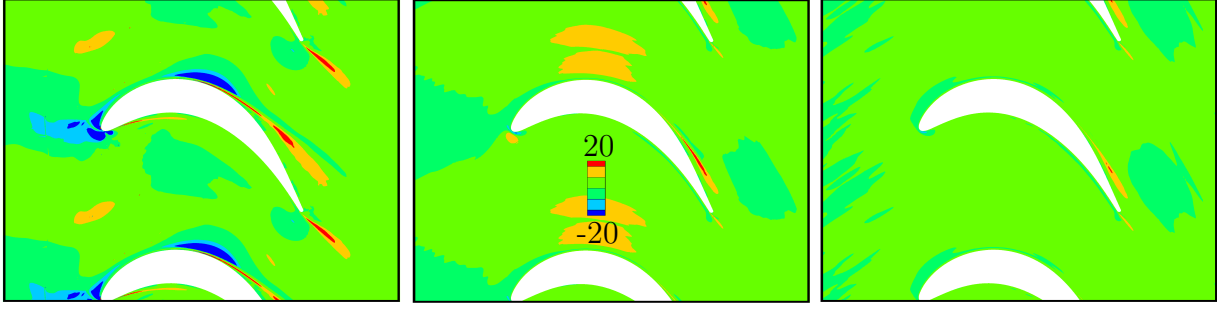


Figure 5.26: Ratio of turbulence production to turbulence dissipation contours at mid-span of MTU-T161 by LES for three free-stream turbulence levels. Left: LES-Clean, Middle: LES-4 %, Right: LES-8 %

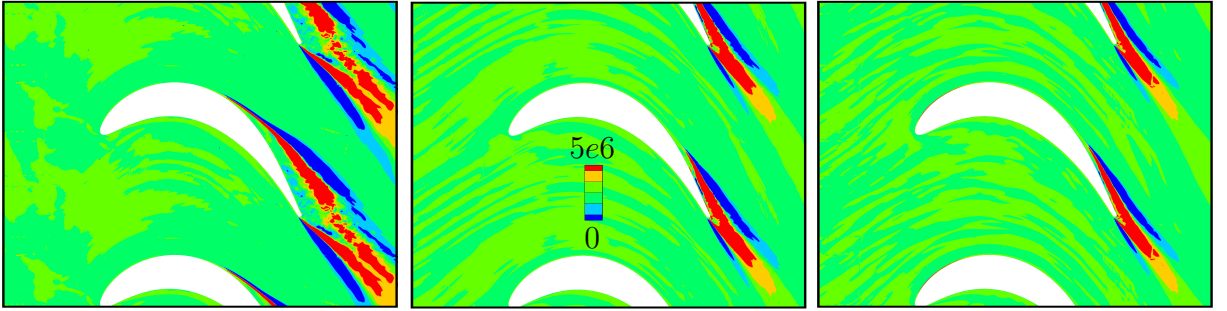


Figure 5.27: Turbulence diffusion contours at mid-span of MTU-T161 by LES for three free-stream turbulence levels. Left: LES-Clean, Middle: LES-4 %, Right: LES-8 %

with free-stream turbulence exhibit elevated convection in the reattachment region. Convection acts as a counterbalance to TKE production within the separated shear layer, redistributing energy across the flow.

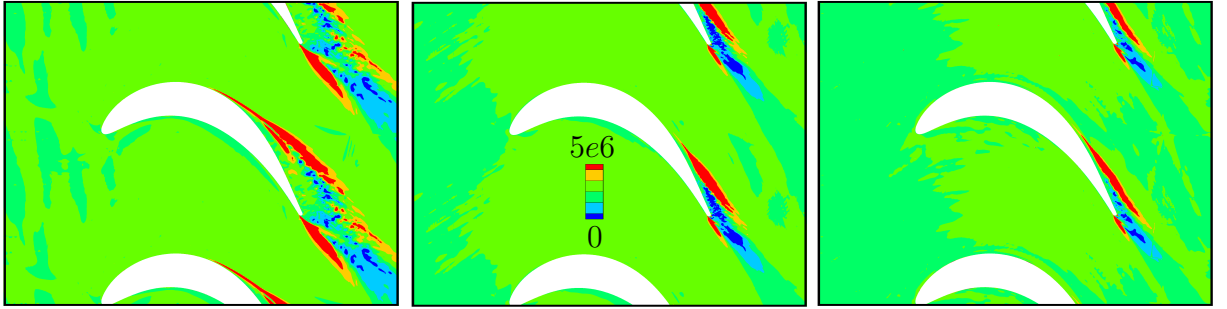


Figure 5.28: Turbulence convection contours at mid-span of MTU-T161 by LES for three free-stream turbulence levels. Left: LES-Clean, Middle: LES-4 %, Right: LES-8 %

The presented results highlight the dynamic interplay between turbulence production, dissipation, diffusion, and convection in the LPT cascade flow under varying free-stream turbulence levels. Additionally, these results emphasize the dominant role of separation bubble in turbulence dynamics. In the separated shear layer, TKE production dominates,

with the production-to-dissipation ratio reaching up to 20, indicating a strong energy imbalance. Diffusion, both turbulent and pressure-driven, plays a critical role in reattachment, particularly in cases with free-stream turbulence, where Kelvin-Helmholtz vortices and shear layer breakdown drive rapid energy redistribution. The LES-Clean case exhibits elevated TKE diffusion and convection along the separated shear layer and wake, reflecting the impact of a non-reattaching separation bubble. Convection counterbalances production in the shear layer, while the absence of reattachment in the LES-Clean case shifts the energy dynamics significantly compared to the free-stream turbulence cases.

5.4 Challenges of RANS in predicting turbine flows

After the validity of the LES results has been demonstrated, the comparison with RANS results has been made, and the LES results have been further analyzed with regard to turbulence behavior, some shortcomings of the RANS methods will be briefly reviewed and summarized in this section.

Boussinesq assumption

The Boussinesq hypothesis corresponds to the linear relation (proportionality) between stress and strain tensors. This proportionality can be easily tested when both tensors are known. In instances where the flow is subjected to periodic strain of relatively high frequency, the phase lag between the stress and strain tensors is of utmost importance. The absence of this feature can result in the continuous over-generation of turbulent kinetic energy. Standard linear eddy-viscosity models have been shown to significantly overpredict the production of turbulent kinetic energy in scenarios involving strong strain. The results in Fig. 5.3 do not show this behavior, because the turbulence model extensions, used in this work, ensure a limitation of TKE-production. The degree of alignment between stresses and strains (C_{as}) can be calculate as follows

$$C_{as} = \frac{a_{ij}S_{ij}}{\|a\|\|S\|} = -\frac{C_{as}}{\|a\|}, \quad (5.1)$$

where the invariants are defined as

$$\|a\| = \sqrt{a_{ij}a_{ij}} \quad \text{and} \quad \|S\| = \sqrt{2S_{ij}S_{ij}} \quad (5.2)$$

The alignment ratio will return a value of 1 when a_{ij} is aligned with S_{ij} and their eigenvectors have equal signs, and delivers a value of 0 when, for instance, there is a lag angle of 45° . This ratio is analogous to the cosine of the angle between vectors. The alignment parameter is designed for utilization within a two-equation modeling framework and quantifies the degree of non-linearity of the stress-strain relationship. In other words, the Boussinesq hypothesis is valid when the ratio is 1. The less it is valid, the more this number is close to 0 (this corresponds to “perpendicular” tensors). Pushing further the analogy with an angle between vectors, one can consider that alignment is approximately verified for angles smaller than $\pi/6$, corresponding to a value of C_{as} larger than 0.86.

Figure 5.29 illustrates the alignment ratio for the RANS simulation of MTU-T161 turbine cascade. The whole flow is dominated by ratios < 0.75 , demonstrating the difficulties of

the RANS turbulence model, used in this chapter, to represent the correct alignment of the stress-strain ratio. In the region of separation and shear layer, the alignment ratio reduces even further.

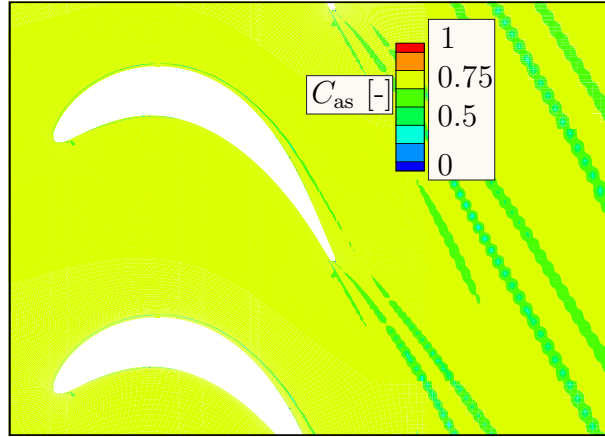


Figure 5.29: Alignment of the shear rate tensor to the anisotropy tensor at mid-span of MTU-T161 based on RANS result

Turbulence Anisotropy

The misalignment between stress and strain tensor, shown in Fig. 5.29, is directly related to the turbulence anisotropy behavior of the flow predicted by RANS turbulence models. As shown in Fig. 5.5, the RANS turbulence model used in this chapter, predicts the turbulence anisotropy very differently (than LES). This is of course a major drawback of the linear eddy viscosity turbulence models, which is inherently due to the modeling assumption used in such models. Fard Afshar et al. (2023a) showed, that the Reynolds-Stress turbulence models (RSM) improve the anisotropy prediction but still have difficulties when compared to LES. This shortcomings are believed to be tackled by modified turbulence models, which are trained to predict the turbulence anisotropy better as in Weatheritt et al. (2017), Akolekar et al. (2018), Akolekar et al. (2019), Zhao et al. (2020a) and Fang et al. (2024).

TKE imbalance

A turbulent boundary layer in equilibrium is a boundary layer where the shape of the boundary layer does not vary significantly in the streamwise direction. In reality, turbulent boundary layers on cascades can behave in a significantly non-equilibrium way. Examples of this include attached boundary layers undergoing transition and reattaching boundary layers, as in MTU-T161 case. Non-equilibrium behavior tends to give rise to a lag between turbulence production and dissipation, as shown in Fig. 5.26. The non-equilibrium turbulence has effects on flow loss prediction by reducing dissipation coefficient, as compared to an equilibrium. At the same time it can also increase flow loss where separation bubbles occur by delaying reattachment. This is important because most turbulence models assume that production and dissipation are largely in balance.

TKE evolution over suction side

As shown in Sec. 5.1.2, RANS model used in this chapter, consistently underpredicts the TKE compared to the LES result. As already discussed before, RANS models use the Boussinesq hypothesis to relate Reynolds stresses to mean flow gradients. During transition, turbulence production is highly non-linear, anisotropic, and intermittent, which

the linear eddy viscosity models are not able to capture correctly. These assumptions lead to an overestimation of turbulence production, which are commonly regulated via additional modifications mentioned before. These modifications reduce the TKE growth in an artificial way. In a physical sense, the boundary layer might remain laminar for longer, and TKE remains lower (than LES) in transitional regions. The shortcoming of eddy viscosity models have been shown in Fard Afshar et al. (2023a) for the LPT case MTU-T161.

Another important factor is the treatment of the transport equation for turbulent kinetic energy (TKE) or Reynolds stresses. In RANS models, the production term is typically based on the assumption of fully developed turbulence, which neglects contributions from early transitional instabilities, such as Klebanoff modes. Additionally, the dissipation rate is often overestimated in transitional regions, suppressing the growth of TKE. The modeling of turbulent transport is also limited, resulting in inadequate diffusion of turbulence into these regions. Collectively, these shortcomings lead to an underprediction of the net TKE growth compared to high-fidelity simulations.

The RANS turbulence model, used in this chapter, aims to fix some of the shortcomings mentioned before by modifying the turbulence production terms such that eddy viscosity is damped in laminar or weakly transitional regions, thereby suppressing the spurious overproduction of TKE that is typically observed in separated shear layers. But the results and comparisons show that these models are unable to overcome these issues properly. Compared to LES, which resolves the anisotropy of turbulent stresses and directly captures the instability-driven growth of fluctuations, viscous blending provides only a modeled correction. In general, the inability of RANS turbulence model to predict the transition is still a key limitation to predict LPT performance. This is despite the advancement of transition models over the last decades. The correct prediction of flow separation and consequently the reattachment, remains another challenge of the RANS turbulence models, as already shown in the current chapter.

5.5 A note on computational cost

In this section, the computational costs of the high-fidelity simulation will be compared with those of the RANS simulation, as this is an important factor for the applicability of such high-fidelity approaches. For this purpose, the computational cost of LES Mesh2 will be compared against RANS, since this mesh was shown in the validation phase to be sufficiently fine to capture the necessary physics. The LES calculation has been conducted on a CPU system using Intel Skylake processors at the high-performance computing (HPC) cluster of the RWTH Aachen university, with a wall clock time of 336 hours on about 1152 CPUs for 40 convective times, which corresponds to around 387,000 core-hours. On the other hand, the RANS calculation is around 100 times faster, requiring only about 3800 core-hours for the calculation. It should be noted that the computational benefit of RANS simulations is typically even higher than 100, since RANS results demonstrate mesh convergence with even coarser meshes. The following comparison shows the efficiency of RANS models and the reason why such methods are still the main tool in the industrial design loop. Nevertheless, as shown in this chapter, such RANS models still struggle to predict the correct behavior in terms of blade loading and flow losses. Despite the high

cost of LES, high-fidelity simulations provide substantially better accuracy in predicting transition, separation, and wake development. Future studies could explore alternative strategies such as zonal hybrid RANS-LES approaches or reduced-order modeling techniques to balance computational cost and accuracy.

6 Numerical Results of a High-Pressure Compressor Cascade

The presented numerical results in this chapter were already validated against measurements in Sec. 4.6., since these are the first numerical analyses of the selected high-pressure compressor cascade based on high-fidelity methods (iLES). Additionally, the iLES results are compared against RANS results to examine the benefits of using the iLES approach over the standard turbulence modeling using the RANS approach. As in the previous chapter, the turbulent behavior of the high-pressure compressor flow will be investigated in terms of anisotropy, entropy, spectral, and quadrant analysis at the center section. The last section of this chapter deals with the influence of the free-stream turbulence on the boundary layer and wake characteristics of the high-pressure compressor flow.

6.1 Time-averaged results at mid-span

In this section, the results of the iLES mesh selected in the last section will be compared with best-practice RANS results. The focus of this section is the comparison of the two methods, in terms of blade surface values, wake flow losses and turbulence analysis at mid-span of the high-pressure compressor cascade profile.

6.1.1 Surface analysis at mid-span

In order to compare the iLES results with those obtained using the RANS methodology, the $p = 5$ iLES results, which demonstrated sufficient accuracy, are used. The $k - \omega$ -SST-model is selected as the best-practice RANS turbulence model, regarding predicting M_{is} over the blade surface and total pressure losses in the wake. The best-practice RANS setup was chosen in preliminary numerical studies involving turbulence model and CFD setup configuration variations. For transition modeling, the $\gamma - Re_\theta$ Menter transition model (Langtry et al. (2009)) is employed. Additionally, the stagnation point anomaly extension by Kato et al. (1993) and a TKE limiter are activated to improve the prediction accuracy of the RANS results.

Blade results

Figure 6.1 compares the normalized isentropic Mach number (M_{is}) and skin-friction coefficient (c_f) distribution over the blade for iLES ($p = 5$), RANS and experimental results. The RANS predictions align well with the iLES results and experiments on the first half of both the suction and pressure sides. However, RANS fails to capture the separation bubble on the suction side (at $x/x_{lax} = 0.5$), a feature predicted by the iLES results.

This inability, similar to the behavior observed in $p = 2$ iLES results, stems from under-resolution due to the low-order discretization inherent in RANS. While RANS predicts the measured pressure distribution on the second half of the pressure side more accurately than iLES, it underperforms on the suction side. Furthermore, the iLES results reveal a small separation bubble on the pressure side near the leading edge, which is not visible in the RANS predictions. It should be noted that the separation bubbles on both the suction and pressure sides are not clearly distinguishable in the averaged c_f distribution (c_f does not cross zero values). However, the pressure side separation bubble becomes evident when examining the averaged streamlines (shown in Fig. 6.1 bottom). Despite the slight underprediction of the M_{is} distribution on the pressure side by iLES, the numerical results match very well with measurements and remain within the measurement error range.

A more precise comparison of these two methods can be achieved by examining the skin-friction coefficient (c_f). Overall, both RANS and iLES show only a few similarities in the c_f distribution over the blade, but significant differences emerge in their predictions of the separation bubble on both the suction and pressure sides. ILES predicts a sharp rise in the skin-friction coefficient on the suction side at $x/x_{lax} = 0.5$ (indication of transition process), a feature absent in the RANS results. The RANS result shows a c_f -peak on the suction side at $x/x_{lax} = 0.1$. A similar discrepancy is observed near the leading edge on the pressure side. In general, there are considerable differences in predicting the c_f distribution over both surfaces by RANS and LES.

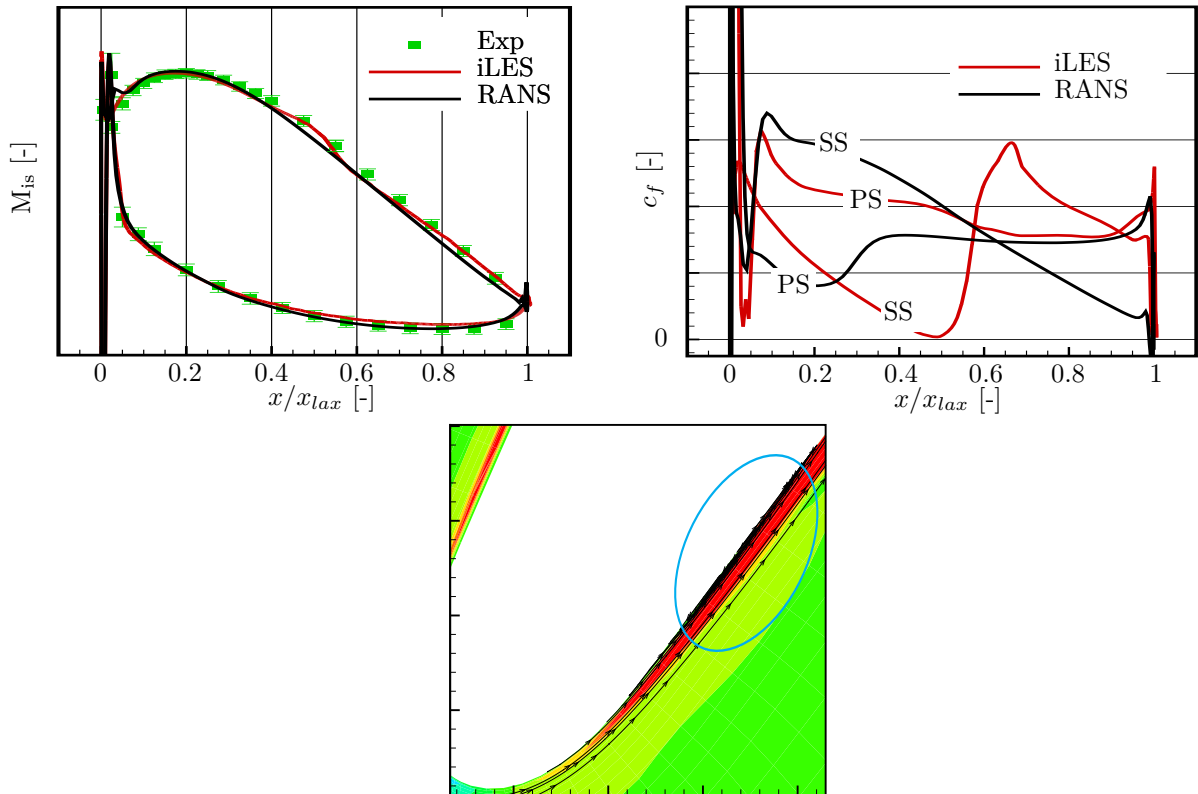


Figure 6.1: Normalized isentropic Mach number distribution (left) and skin-friction coefficient distribution (right) over the high-pressure compressor blade by iLES and RANS. Bottom: Streamlines in the front part of the pressure surface, predicted by iLES.

The deficiencies of the RANS model to accurately predict the pressure distribution may be traced back to the inadequate reproduction of the turbulence effects in the transitional and separation region. Since the separation bubble is in the transitional state, the turbulence is mostly generated at the outer part of the separation bubble, where strongly anisotropic characteristics are present, as will be shown later. These characteristics are also propagated into the wake. In order to analyze the aforementioned hypothesis, the turbulence behavior at the mid-span section of the high-pressure compressor will be examined for both LES and RANS in the next section. The TKE analysis, in the next sections, will be presented as contour plots in the next sections, since it is not possible to conduct such comprehensive analyses via measurements. This emphasizes the advantages of high-fidelity simulations.

6.1.2 TKE analysis at mid-span

This section focuses on analyzing the turbulence behavior at the mid-span of the high-pressure compressor blade, with a particular focus on the suction side of the high-pressure compressor cascade. To achieve this, the turbulent kinetic energy (TKE) contours and profiles obtained from both iLES and RANS simulations are examined.

The normalized TKE distribution at mid-span, as shown in Fig. 6.2, highlights the differences between iLES and RANS predictions, particularly on the second half of the suction side and in the near-wake region. ILES predicts higher TKE levels than RANS across the domain at mid-span. However, RANS shows an earlier rise in TKE levels over the suction side (marked in Fig. 6.2 as a black circle), leading to premature turbulence and suppressing the flow separation as discussed in Fig. 6.1. Both methods reveal elevated TKE levels in the latter part of the suction side and near the trailing edge.

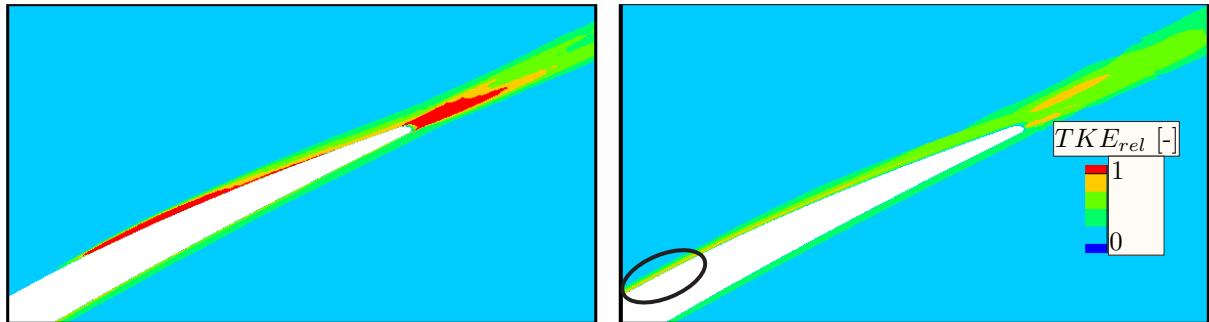


Figure 6.2: Normalized turbulent kinetic energy by iLES (left) and RANS (right). TKE values are normalized with the maximum values of iLES.

The turbulence production, P_{TKE} (Fig. 6.3), on the other hand, shows higher values for the RANS model over the entire pressure- and suction-side. In RANS formulation, the turbulence production is derived based on the velocity gradients and the Reynolds stresses. However, high turbulence production does not necessarily translate into high TKE because of the balance between production, dissipation, and transport terms (cf. (2.43)). In RANS results, there are high turbulence production values observed, because the turbulence model responds to high velocity gradients in the boundary layer, especially near regions of strong adverse pressure gradients. However, much of this production is dissipated or

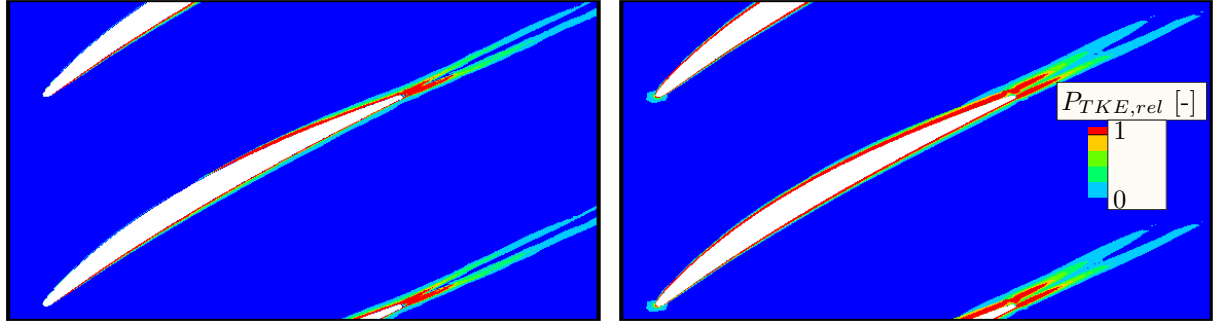


Figure 6.3: Normalized turbulence production by iLES (left) and RANS (right). P_{TKE} values are normalized with the maximum values of iLES

redistributed by the turbulence model. However, iLES shows very similar results for P_{TKE} as for TKE.

Figure 6.4 further investigates the TKE evolution along the suction side and confirms the findings in Fig. 6.2 and Fig. 6.3. RANS predictions show higher TKE values up to $x/x_{lax} = 0.4$, with an overproduction of TKE reaching up to seven times that of iLES. The positions of the TKE peaks are consistent between the two methods up to this point. Starting from $x/x_{lax} = 0.5$, the iLES result indicates a sharp increase in TKE, surpassing RANS predictions and remaining higher along the rest of the suction side. In the second half of the suction side, RANS underpredicts TKE levels by up to three times compared to iLES. Moreover, the second half of the suction side shows slight positional shifts (wall normal) in the TKE peaks between iLES and RANS. iLES captures more distinct and sharper TKE peaks, reflecting its ability to resolve finer turbulence structures. In contrast, RANS provides smoother profiles, which can obscure localized turbulence phenomena. The results of Fig. 6.4 are in agreement with the contours shown in Fig. 6.2, where iLES predicts high TKE values only after the flow separation. RANS overpredicts the TKE in the acceleration region of the suction side surface and prevents the flow separation.

Figures 6.2 and Fig. 6.4 illustrate the distinct treatment of turbulent kinetic energy (TKE) by iLES and RANS over the suction side of the compressor cascade. It is important to note that the comparison involves resolved TKE from iLES and modeled TKE from RANS. However, the contribution of modeled TKE in iLES is negligible. The iLES and RANS results of the compressor cascade flow, regarding the TKE evolution, differ from the turbine cascade results (cf. Sec. 5.1.2). This difference can be attributed to the differences in flow physics and boundary layer characteristics of the two types of cascades. Additionally, different RANS turbulence models have been utilized for each case, and therefore the turbulence is modeled and modified slightly differently. These differences underline the inherent limitations of RANS and the strengths of iLES for accurately resolving complex flow phenomena. Further analysis of the comparison between these methods is detailed in the following sections.

6.1.3 Total pressure loss at wake plane

Figure 6.5 illustrates the total pressure loss in the wake area (three axial planes) for both iLES and RANS simulations, alongside experimental results. The iLES predictions align

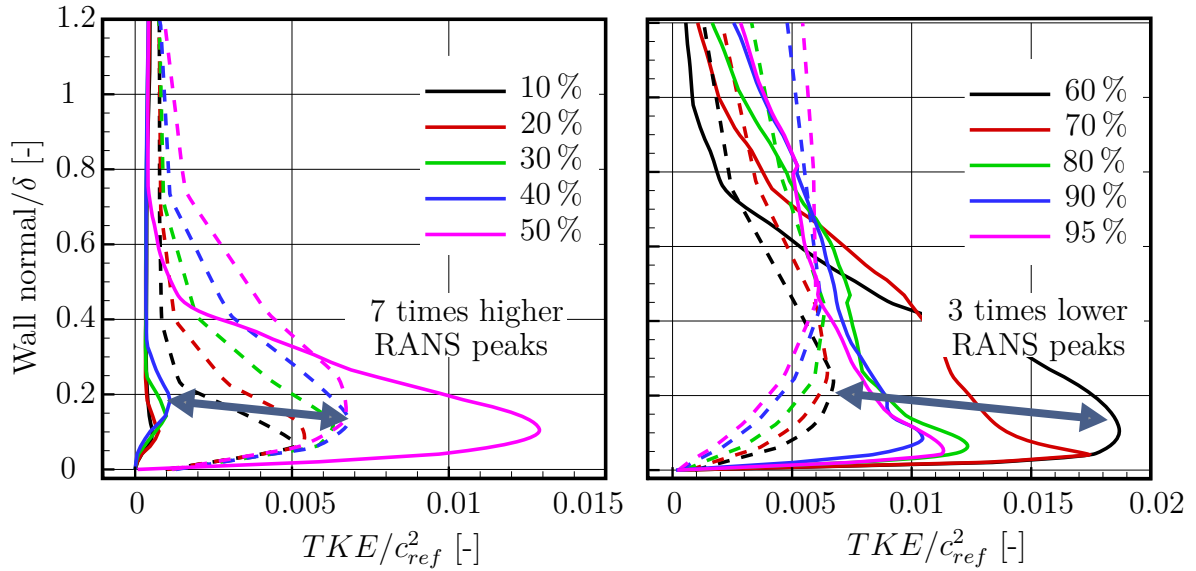


Figure 6.4: Evolution of normalized TKE over the suction side by RANS (dashed) and iLES (solid). Left: Boundary layer cuts over the first half of the suction side. Right: Boundary layer cuts over the second half of the suction side.

very well with the experimental data across all three wake planes, accurately capturing both the wake flow loss peak and width of the wake flow loss distribution, as already shown in Sec. 4.6.1. The RANS results, on the other hand, overestimate the wake flow losses, both in terms of peak magnitude and width, which is a well-known limitation of RANS methods in flow loss prediction. However, RANS does reasonably capture the decay of wake flow losses as the flow progresses downstream. The integral flow loss deficit between the RANS and iLES results varies between 45 % and 55 % for different axial wake planes. These observations underscore the advantages of high-fidelity iLES in resolving complex flow features like separation bubbles and wake flow losses, while highlighting the persistent limitations of RANS in such scenarios.

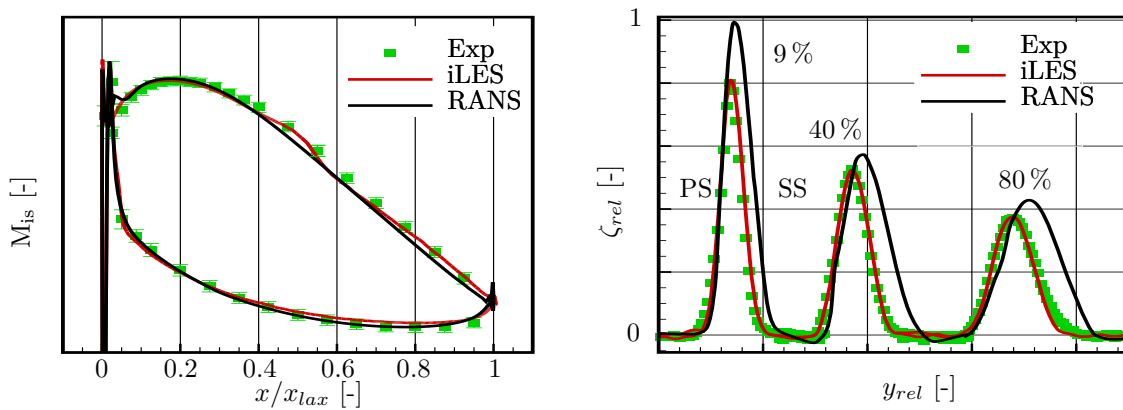


Figure 6.5: Normalized isentropic Mach number distribution over the blade and normalized total pressure loss of high-pressure compressor blade in wake by iLES, RANS and experiments.

6.2 Further Analysis of LES results

In order to better understand the results and discrepancies between RANS and iLES, the turbulence behavior of the iLES results will be analyzed in further detail in the current section.

6.2.1 Anisotropy

Another important aspect of turbulence analysis is the degree of turbulence anisotropy predicted by the respective simulation methods, as discussed in the previous chapter. Figure 6.6 presents the componentality of the Reynolds stress anisotropy tensor as predicted by both the iLES and RANS approaches. The comparison highlights notable differences in anisotropy behavior within the passage flow between the two methods.

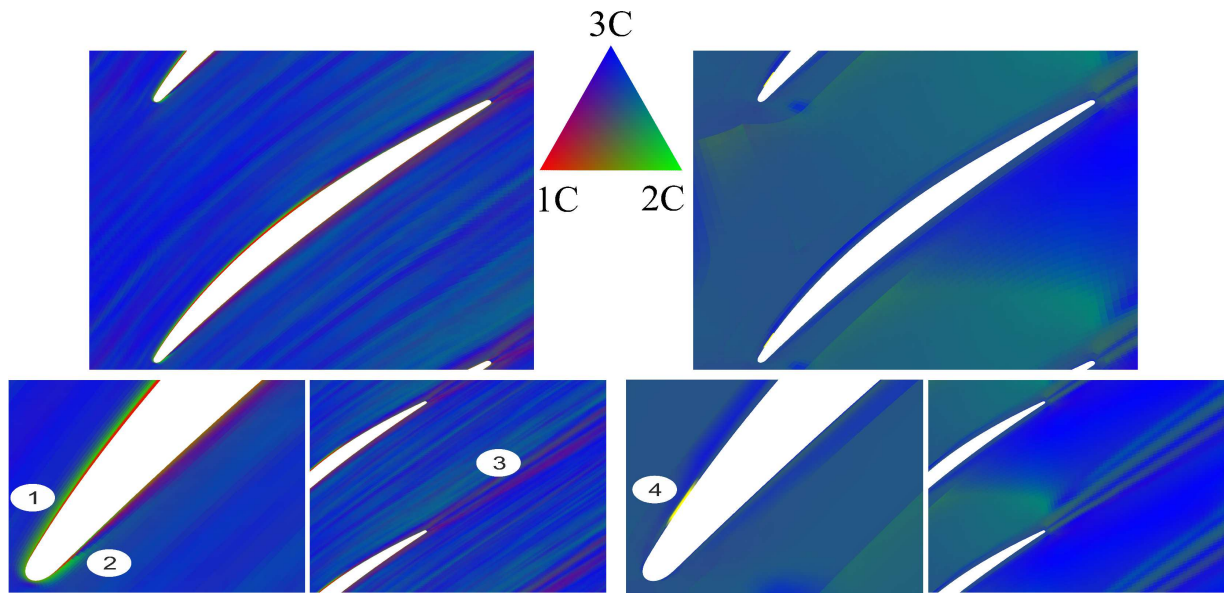


Figure 6.6: Componentality of Reynolds stress anisotropy tensor for iLES (left) and RANS (right). The colors of the contours are described in Sec. 2.2.1.

In the passage flow, the iLES results predominantly exhibit blue regions, indicating areas of three-component (3C) turbulence. In contrast, the RANS predictions reveal a more heterogeneous distribution, characterized by a mixture of two-component (2C, green) and 3C turbulence. The most pronounced discrepancies occur within the boundary layer and wake regions. In particular, iLES captures a thin layer of one-component (1C, red) turbulence in close proximity to the wall, which transitions into a region of 2C turbulence as the distance from the wall increases. This behavior is especially evident on the suction side of the blade (denoted as Region 1 in Fig. 6.6), where the entire surface is characterized by this highly anisotropic turbulence structure. On the pressure side, 2C turbulence is largely confined to the leading edge region (Region 2), and the presence of 1C turbulence is less prominent compared to the suction side. Additionally, the spatial distribution and interaction between 2C and 3C turbulence on the pressure side differ slightly from those observed on the suction surface. It is observed that there is a substantial similarity in the anisotropic behavior between high-pressure compressor flows and low-pressure turbine

flows. It is particularly important to note the similarities in the leading-edge area and the way in which the turbulence behaves at the leading edge.

In the wake region, iLES identifies two distinct peaks of 1C turbulence located near the trailing edge (Region 3). As will be demonstrated later, these peaks correspond to regions of elevated turbulence production downstream of the trailing edge, originating from the pressure and suction side flows, respectively. In contrast, the RANS simulations predict a fundamentally different anisotropy distribution within the boundary layers. Specifically, RANS fails to capture the distinct 1C turbulence layer adjacent to the wall surfaces on both the suction and pressure sides. Additionally, the 2C turbulence region is not as distinctly defined in the RANS results. Instead, RANS predicts a localized region near the suction side (Region 4) where turbulence anisotropy is characterized by a blend of 1C (red) and 2C (green) turbulence. In the wake region, RANS results qualitatively agree with the iLES prediction in identifying two peaks of anisotropic turbulence. However, these peaks appear less distinct in terms of spatial resolution and intensity. While iLES predicts localized 1C turbulence (red regions), RANS identifies the same areas as 2C turbulence (green regions), indicating a lower degree of turbulence anisotropy captured by the RANS model.

Overall, the anisotropy analysis reveals that RANS and iLES not only differ in their predictions of the magnitude and evolution of the turbulent kinetic energy (TKE), but also in the detailed structure of turbulence composition. These discrepancies are largely attributed to the limitations of the linear eddy-viscosity hypothesis and the Boussinesq approximation inherent in conventional RANS models. As demonstrated in Fig. 6.7, the turbulence behavior predicted by each method (iLES and RANS) along a streamline over the blade is illustrated. The iLES reveals that along a streamline, the flow encounters nearly all turbulence states (1C, 2C and 3C). On the other hand, the $k - \omega$ -SST-model predicts turbulence behavior as either isotropic or plain strain.

6.2.2 Entropy analysis

The following section presents an entropy analysis of the mid-span flow in the high-pressure compressor cascade using both iLES and RANS, consistent with the approach used in the previous chapter. The goal is to identify the regions with the highest entropy generation rate and the associated flow losses. The focus will be on two terms in particular, the mean viscous dissipation (V_M) and the Reynolds stress production (P_R) terms. These terms have been shown to contribute significantly to entropy generation in different regions. The results of A_{TKE} are not shown here, because its contribution to the overall entropy generation is not as prominent as the other two terms.

Figure 6.8 illustrates the entropy generation associated with turbulence (Reynolds stress) production for both iLES and RANS. The iLES results reveal high levels of turbulence production-related entropy generation in the suction-side boundary layer, pressure-side boundary layer and the wake stream. On the suction side, the turbulence-related entropy generation starts from the leading edge, continues along the separated shear layer and becomes mostly prominent from the mid-blade region, corresponding to the laminar-turbulent transition. This figure confirms the role of the transition in turbulence production and the associated losses. The entropy generation is initiated at the leading edge of

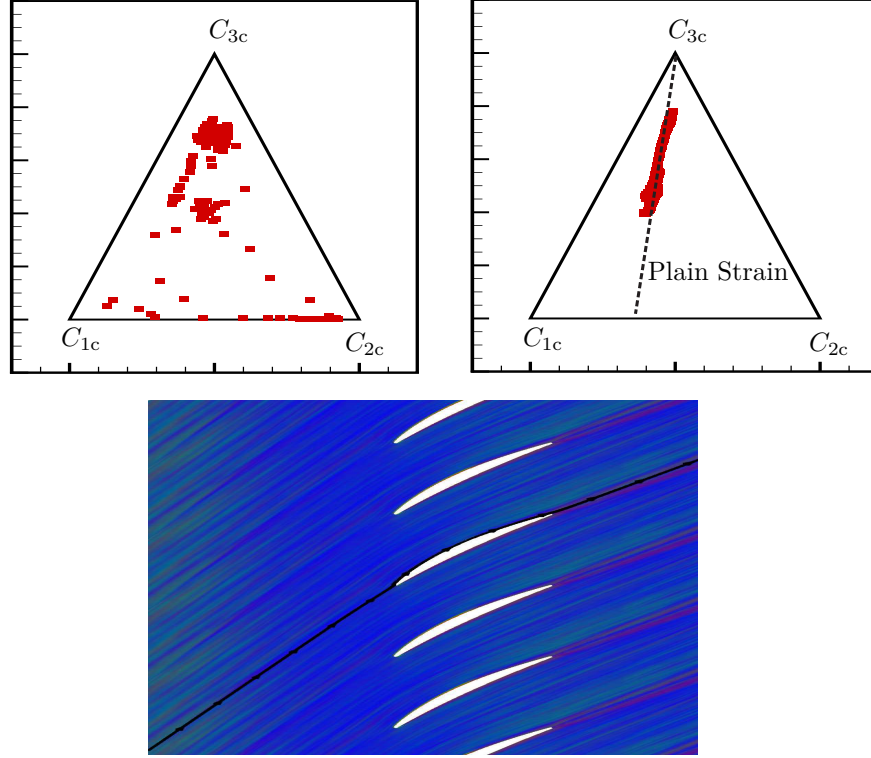


Figure 6.7: Barycentric analysis of turbulence along a streamline shown in bottom row. Left: iLES, Right: RANS

the pressure-side surface and persists over the entirety of the surface area. In the wake region, two distinct peaks are observed, corresponding to the pressure-side and suction-side boundary layer flow. These peaks are related to vortex shedding downstream of the trailing edge. These findings align with and are directly linked to the overall TKE distribution shown in Fig. 6.2.

The RANS results show some similarities with iLES regarding entropy generation. RANS also predicts turbulence-related entropy to be primarily generated in the boundary layers of the suction side, pressure side, and the wake stream. However, RANS predicts entropy generation starting at the leading edge and extending across the entire suction side. The overproduction of the P_R term is associated with the inability of RANS to effectively model turbulence production at stagnation points. A similar pattern is observed on the pressure side, although the predictions show more resemblance to the iLES results in this region. Furthermore, RANS captures two distinct peaks in the wake region, as seen in iLES, but with slightly different spatial distribution. In general, RANS predicts a thicker region that is associated with entropy generation. These findings support the overestimation of wake flow losses by RANS, as demonstrated in Fig. 6.5. At the first axial wake plane (in Fig. 6.5), the flow loss predictions between iLES and RANS show considerable differences. However, these differences diminish further downstream. This reduction in discrepancies can be attributed to the convergence of entropy generation by the two methods as the flow progresses downstream.

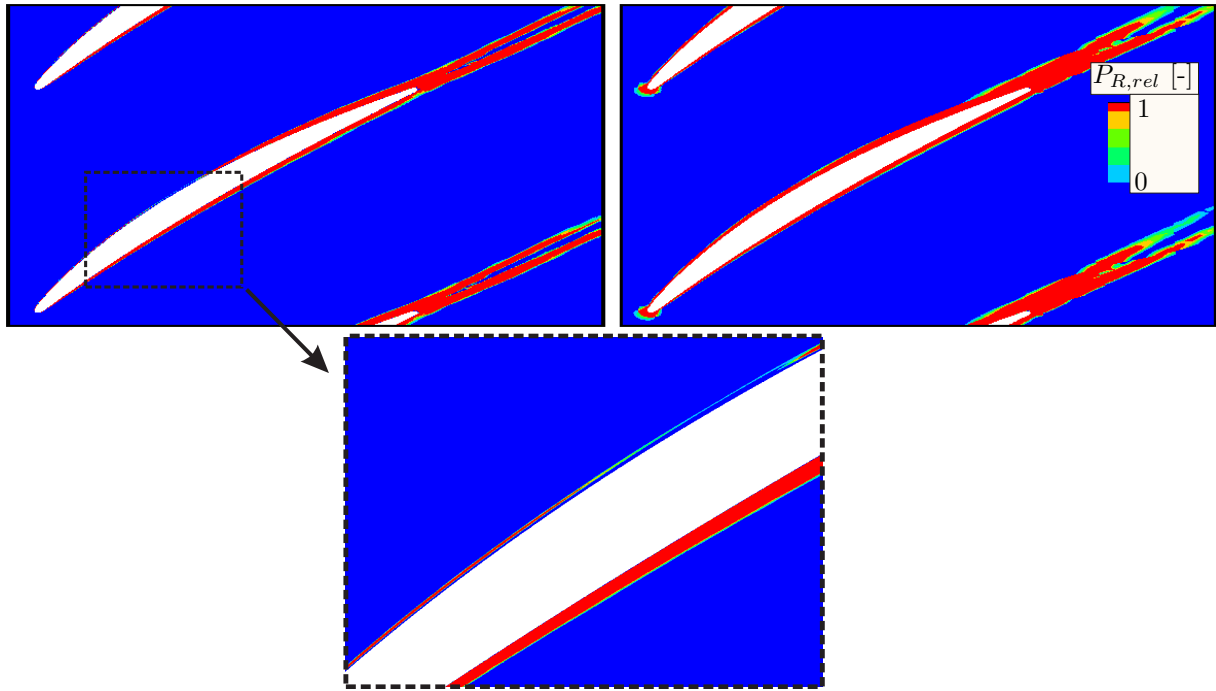


Figure 6.8: Normalized entropy production rate, related to turbulence production, cf. (2.103) and cf. (2.102) for iLES (left) and RANS (right)

Figure 6.9 illustrates the entropy generation associated with mean viscous dissipation (cf. (2.102)) for both iLES and RANS. The results show a strong similarity between the two methods, with only minor differences observed. These differences, noticeable at the trailing edge and along the second half of the suction-side profile, are highlighted in the zoomed version of Fig. 6.9 (bottom row). From the entropy contour plots, it can be concluded that the differences in entropy generation, and consequently the associated flow losses, between iLES and RANS are primarily driven by turbulence production on the suction side. The effects of the mean viscous flow are mainly confined to the blade boundary layer, while the turbulence effects originate from the turbulent boundary layer on the suction side and vortex shedding in the wake region.

Following the comparison of numerical methods in terms of entropy generation, as evidenced by the examination of contours at the mid-span plane (with a primary focus on the suction side), the focus has now shifted to total entropy generation, with a particular emphasis on the contribution of turbulence-driven entropy production (P_R) at the wake planes (cf. Fig. 6.10).

To assess the entropy generation at the wake planes fairly, the overall entropy generation (G_s) will first be considered. The left column of Fig. 6.10 shows the overall entropy generation for iLES and RANS at all three wake planes. Both iLES and RANS results show two peaks, corresponding to turbulence production on the suction and pressure sides. These peaks are observed across all three wake planes but become less pronounced at downstream wake planes. In general, iLES predicts higher overall entropy generation peaks than RANS (both positive and negative), but with high fluctuations related to the convergence of this term. The negative values of G_s mostly result from the negative values of A_{TKE} (not shown here). The elevated levels of negative and positive G_s values, as predicted by LES, may provide a rationale for the reduced overall integral total pressure

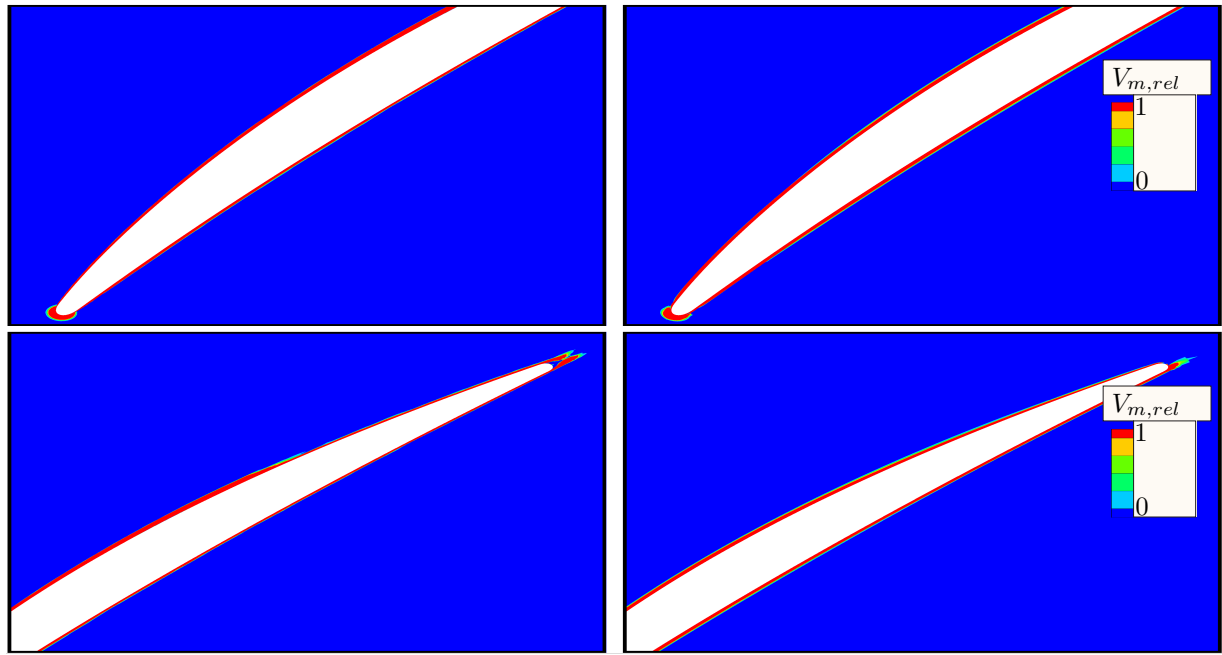


Figure 6.9: Normalized mean viscous dissipation term of entropy generation rate cf. (2.103) and cf. (2.102) for iLES (left) and RANS (right)

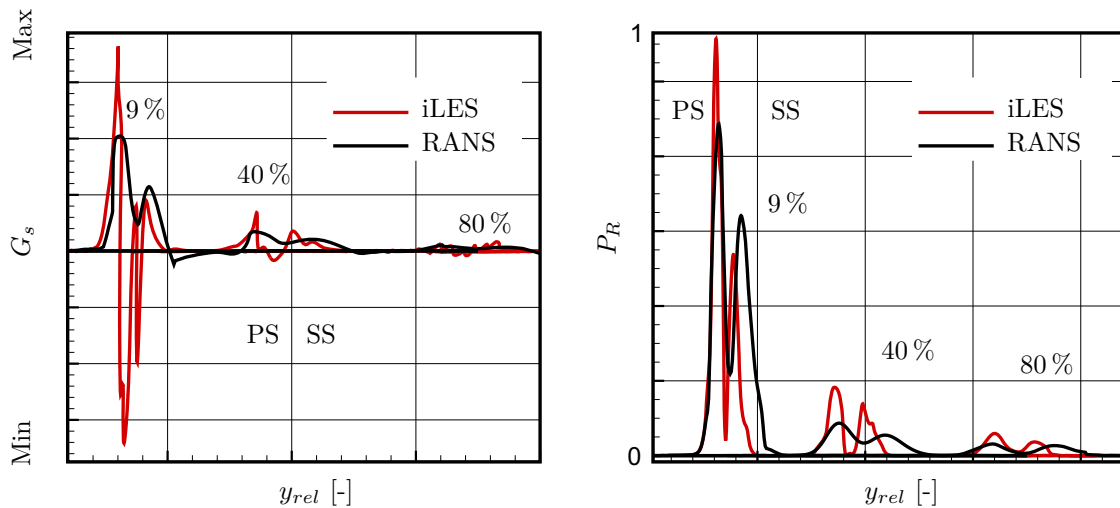


Figure 6.10: Normalized overall entropy production rate cf. (2.103) and cf. (2.102) (left) and normalized entropy production rate cf. (2.103) and cf. (2.102) (right) in wake for iLES and RANS

loss at wake planes by LES. Furthermore, the decay of G_s values at the wake planes aligns closely with the decay of total pressure loss at the same locations.

The right column of Fig. 6.10 displays the entropy associated with turbulence production at the three wake planes for both iLES and RANS. Both iLES and RANS results show two peaks corresponding to turbulence production on the suction and pressure side boundary layer flow. These peaks are observed across all three wake planes but become less pronounced as they move downstream. The entropy predicted by iLES is higher than

that predicted by RANS, but with positive and negative values. The overall picture of P_R distribution at all three wake planes is very similar to G_s distribution. It is important to note that the turbulence production term (P_R) represents the energy transferred from the mean flow to turbulent fluctuations. As such, P_R can take negative values in iLES, which indicates a scenario where the turbulent kinetic energy (TKE) inversely cascades into mean kinetic energy. Such a scenario is not permitted by RANS.

6.2.3 Spectral analysis

In the current section, the spectral behavior of the flow on the suction and pressure sides of the high-pressure compressor profile mid-span will be discussed, using the Welch method introduced earlier (cf. Sec. 2.8.1.2). For this purpose, the time series data of the iLES, stored in probes at different boundary layer cuts over the suction and pressure sides, are used. To visualize the power spectral density (PSD) in more detail, the suction side figures focus on the inner layers of the boundary layer near the wall, up to a wall probe distance of 8 % of maximum probe normal distance to the wall, at each boundary layer cut. This choice is made due to the thinner boundary layer associated with the higher Reynolds number operating point, compared to the results in Sec. 5.2.3.

Figure 6.11 shows the PSD of the streamwise velocity (velocity along the suction side surface) for the first half of the suction side. The positions of the probes, as relative axial chord length, and the TKE contour are indicated at the bottom of Fig. 6.11. At the first boundary layer cut near the leading edge, there are high-energy velocity fluctuations over a broad wall-normal distance, primarily due to the interaction of the free-stream turbulence and the stagnation point. Nevertheless, shortly after the first boundary layer cut at $x/x_{lax} = 0.1$, most of the energy related to the streamwise velocity fluctuations is concentrated near the wall. This area corresponds to the pre-transitional region. The spectral analysis of the pre-transitional area in the LPT cascade MTU-T161 showed a similar behavior (cf. Sec. 5.2.3), with less noise. The spectral analysis of the boundary layer cuts at $x/x_{lax} = 0.1$ and $x/x_{lax} = 0.22$ represents and confirms the concept of shear sheltering introduced in Sec. 2.6.1. As the flow moves downstream towards the middle of the suction side, the regions containing the most energy remain near the wall but begin to lift off the wall (compared to $x/x_{lax} = 0.1$). This behavior is related to the transition mechanism shown in Fig. 6.5 and was also observed in the spectral analysis of LPT cascade MTU-T161. The region of transition to turbulence ($x/x_{lax} = 0.52$) exhibits the maximum energy level. As in the previous figures, the scales in the colorbars are set individually for each plot.

Overall, the spectral analysis of the first half of the suction side shows that the areas with the most energetic fluctuations are primarily confined to the wall-normal regions below 2.5% of the normalized wall distance. The frequency spectrum of the energy-containing fluctuations is mainly limited to $ft_c < 1$ (indication of time scales larger than a convective time unit). In the second half of the suction side flow (cf. Fig. 6.12), a shift in the frequency spectrum to higher ft_c values is observed, similar to what was seen in the LPT cascade flow (cf. Fig. 5.11). At all axial positions, the maximum values of the PSD increase, compared to the first half. This behavior is attributed to the higher energy content in the fully turbulent flow state. At $x/x_{lax} = 0.58$, a similar behavior to the axial cut at $x/x_{lax} = 0.52$ is observed, but with the energy-containing region closer to the wall, indicating a possible

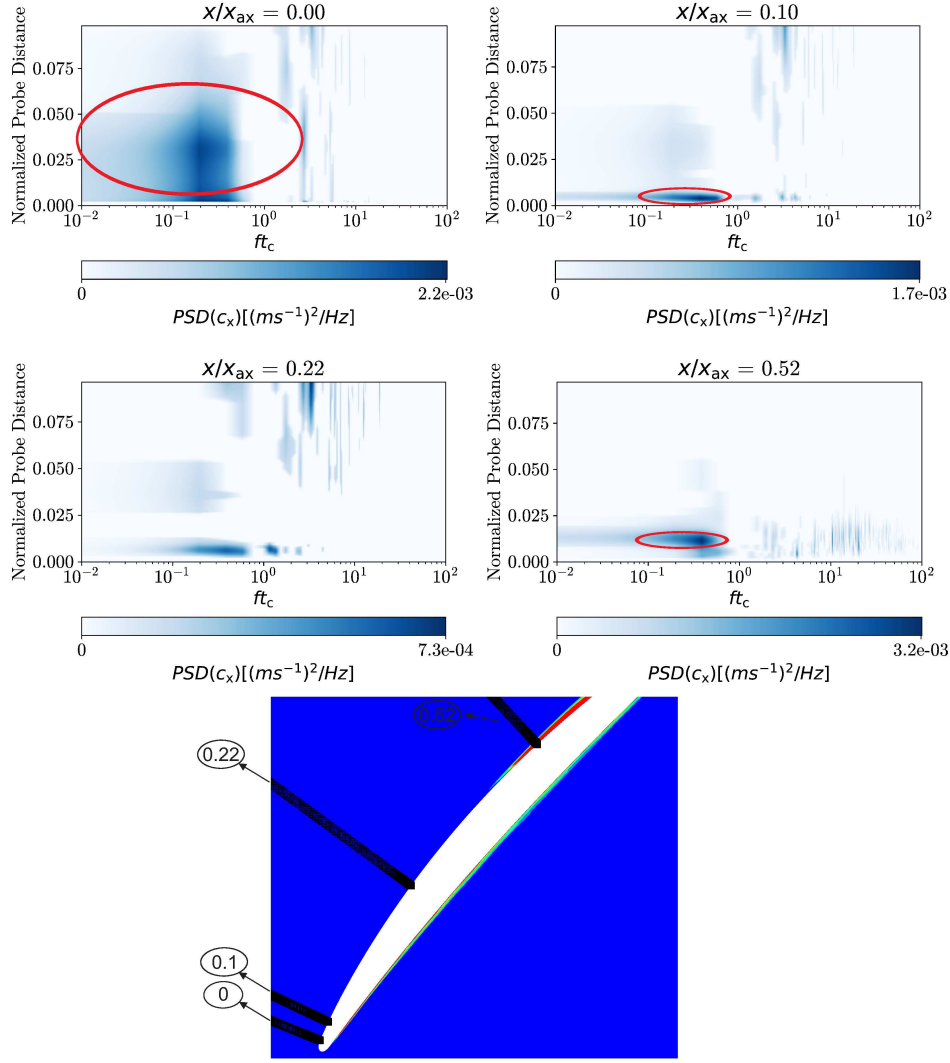


Figure 6.11: PSD contour of streamwise velocity over frequencies for different axial positions over the first half of the suction side based on iLES. Zoomed blade-to-blade view and the positions of the corresponding boundary layer cuts with TKE contour. The PSD colorbars are scaled individually.

reattachment. Starting from the axial position, $x/x_{lax} = 0.64$, velocity fluctuations with a broader spectrum and peaks begin to contribute to the energy bounded in the boundary layer region. After reaching the final stages of fully turbulent flow at $x/x_{lax} = 0.88$ and $x/x_{lax} = 0.94$, no layer parallel to the wall containing higher velocity fluctuation energy is observed.

Figure 6.13 shows the PSD of the streamwise velocity (velocity along the pressure side surface) for the first half of the pressure side. At the bottom of the figure, the positions of the probes and the TKE contour are indicated. Near the leading edge, high-energy velocity fluctuations extend over a broad wall-normal distance, which are primarily related to the interaction of the free-stream turbulence and the stagnation point. This behavior is similar to the one observed on the suction side, with fluctuations concentrated at $ft_c < 1$. Downstream to the first axial position, from the axial position $x/x_{lax} = 0.07$ to $x/x_{lax} = 0.32$, the fluctuation energy is spread over a frequency range of $0.1 < ft_c < 100$ and remains

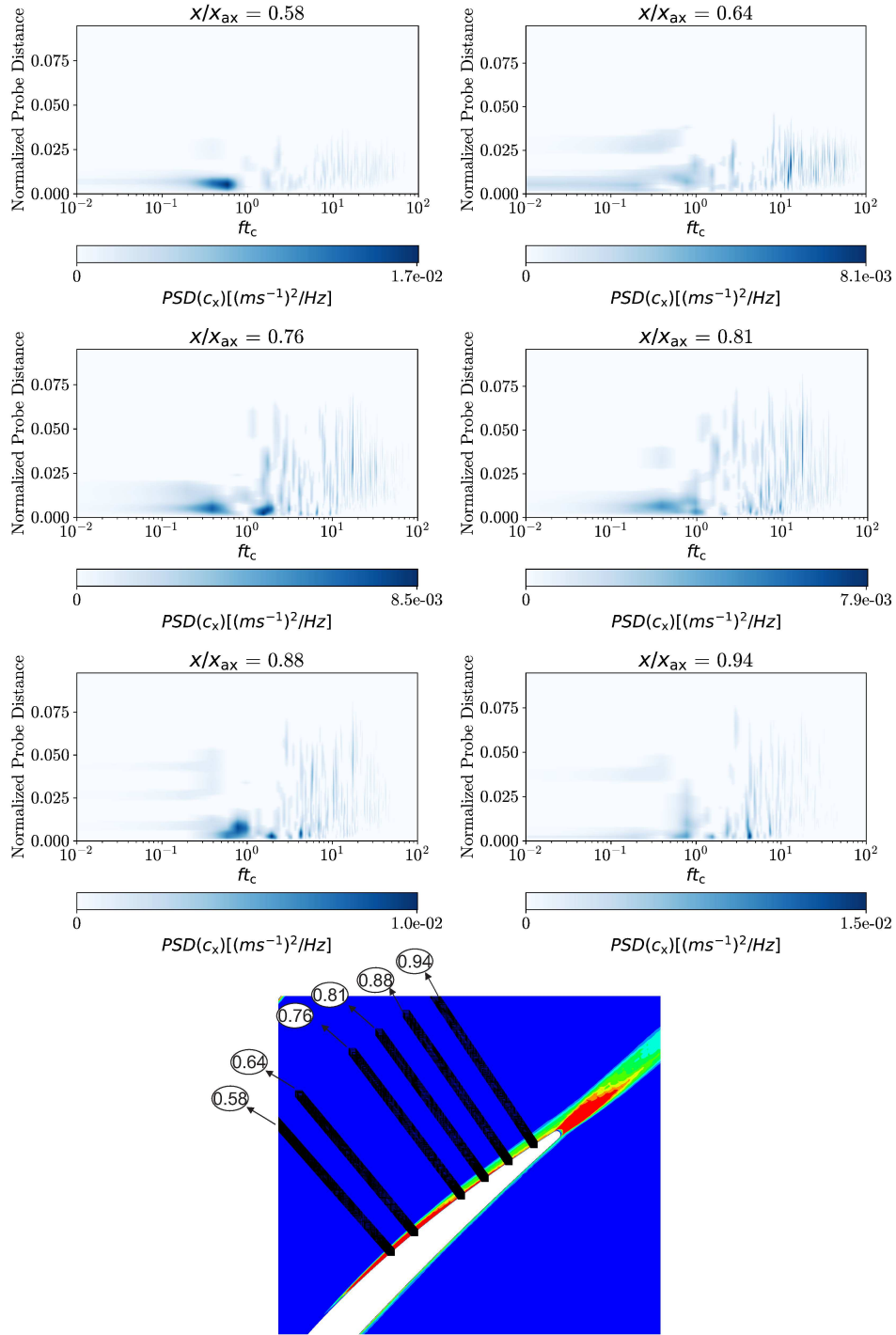


Figure 6.12: PSD contour of streamwise velocity over frequencies for different axial positions over the second half of the suction side based on iLES. Zoomed blade to blade view and the positions of the corresponding boundary layer cuts with TKE contour. The PSD colorbars are scaled individually.

confined near the wall. At $x/x_{lax} = 0.32$, the fluctuations begin to elevate from the wall. By the time the flow reaches the middle of the chord, the PSD energy of the fluctuating velocity becomes distributed across a wide spatial and temporal range.

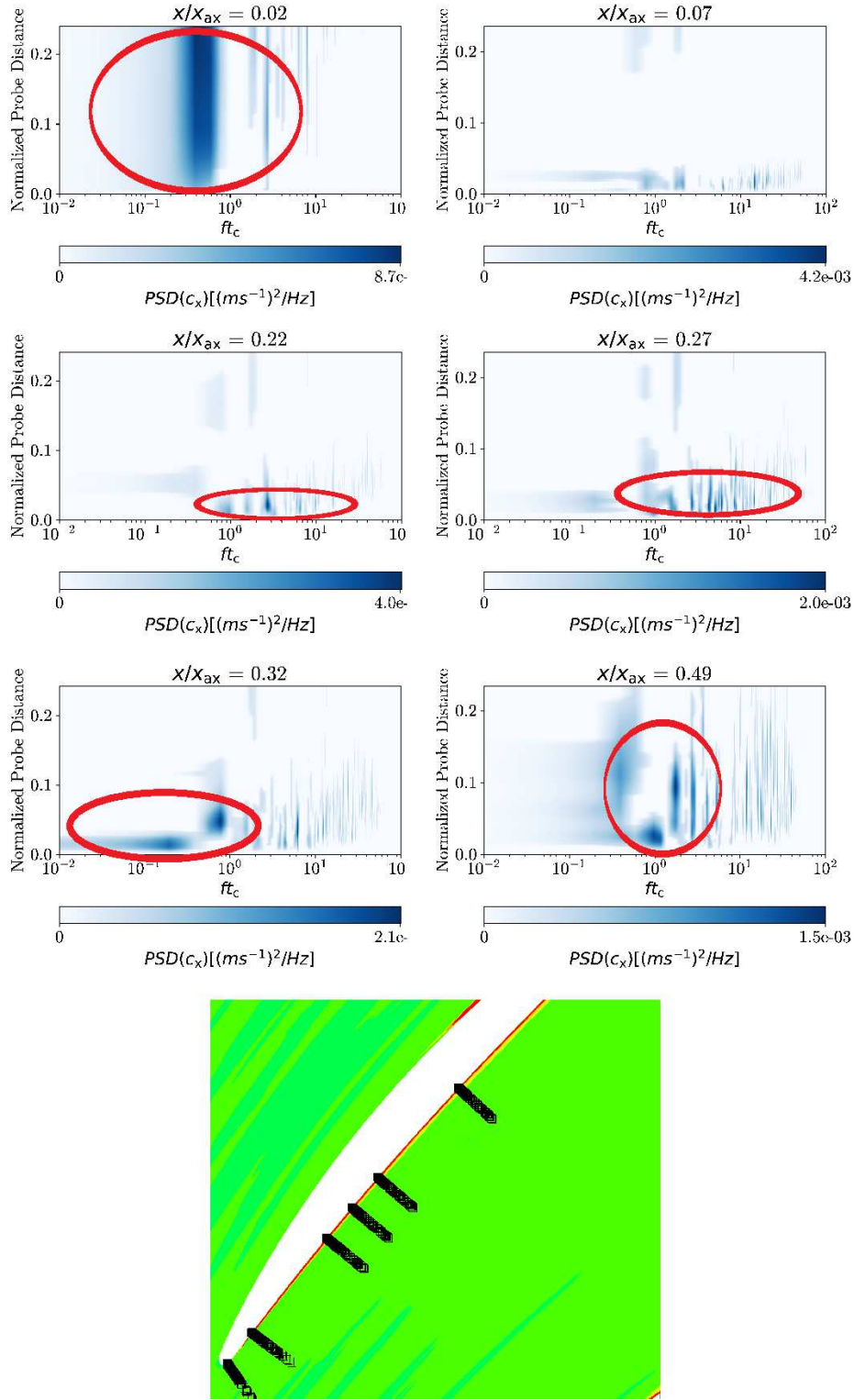


Figure 6.13: PSD contour of streamwise velocity over frequencies for different axial positions over the first half of the pressure side based on iLES. Zoomed blade to blade view and the positions of the corresponding boundary layer cuts with TKE contour. The PSD colorbars are scaled individually.

In the second half of the pressure side, the fluctuation behavior becomes more dispersed across a broader frequency range, particularly for $ft_c > 1$. At axial positions $x/x_{lax} = 0.54$

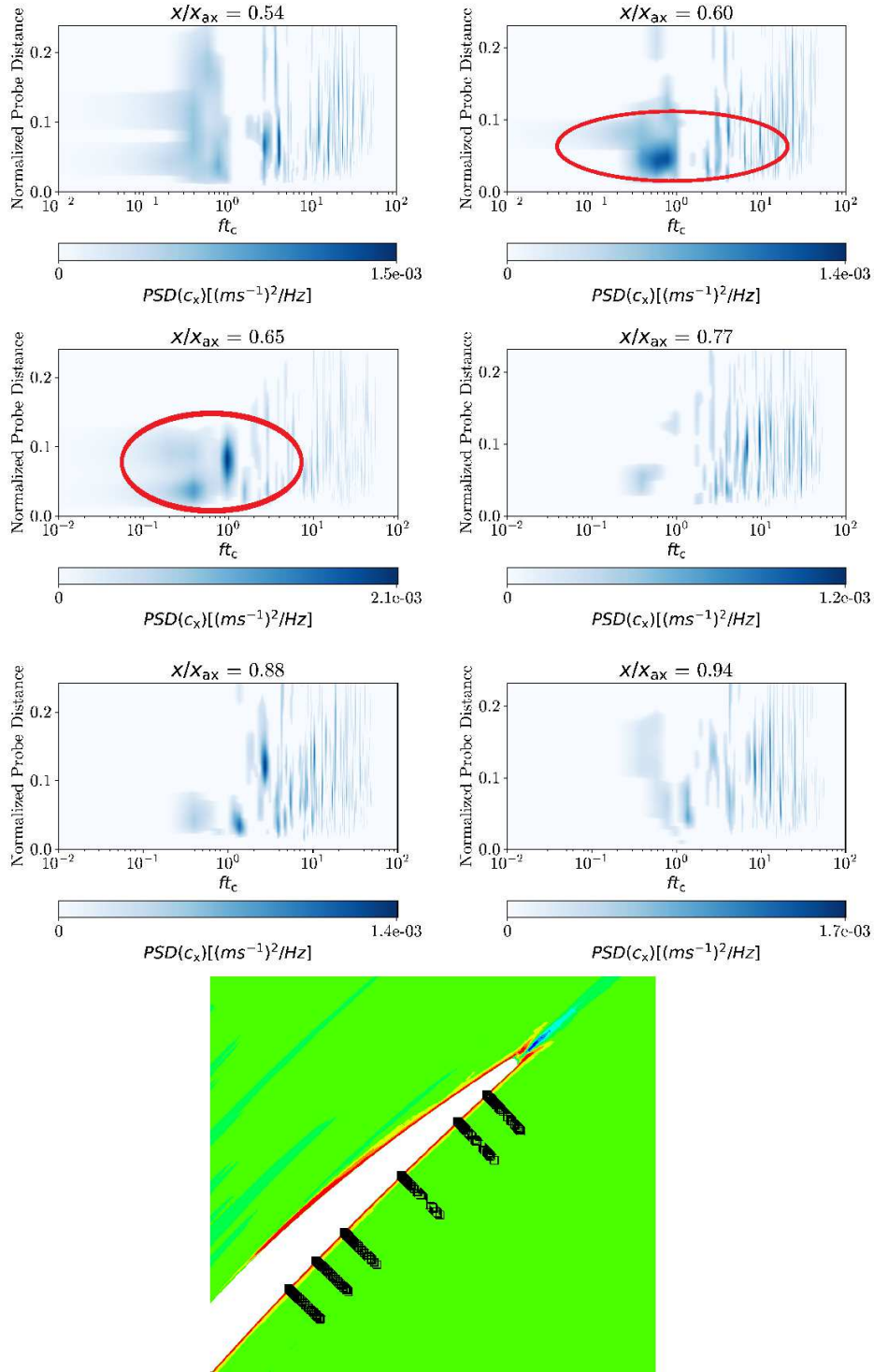


Figure 6.14: PSD contour of streamwise velocity over frequencies for different axial positions over the second half of the pressure side based on iLES. Zoomed blade to blade view and the positions of the corresponding boundary layer cuts with TKE contour. The PSD colorbars are scaled individually.

and $x/x_{lax} = 0.60$, there is still some energy content in the low-frequency range $ft_c < 1$. However, moving into the last third of the pressure side, the energy content shifts almost

entirely to $ft_c > 1$, and the spectral behavior becomes more uniform over the entire wall normal spatial range.

Overall, many similarities exist between the spectral behavior of velocity fluctuations on the pressure and suction sides. Based on these results, it can be concluded that near the leading edge, the free-stream turbulence or velocity fluctuations are predominantly distributed across the passage or over a broader wall-normal distance, with frequency content primarily at $ft_c < 1$. Following a brief axial development, the velocity fluctuations enter the boundary layer and begin to elevate from the wall as transition or flow separation begins. The results obtained from the PSD analysis lend further support to the hypothesis that the boundary layer functions as a low-pass filter. In the final stage of transition, where the flow becomes fully turbulent, the energy shifts predominantly to higher frequencies, and the peaks spread over a wider spatial range.

6.2.4 Quadrant analysis

The quadrant analysis of the Reynolds stress tensor is performed for distinct axial boundary layer positions at the mid-span of the high-pressure compressor cascade to identify the principal contributions to turbulence productions based on Reynolds stresses. In the case of the high-pressure compressor, the focus is on different axial positions than LPT cascade MTU-T161, since these positions are more representative for the transitional and fully turbulent regions. As in the previous quadrant figures, at each boundary layer section five positions normal to the suction side surface are selected to plot the contribution of each quadrant to the Reynolds shear stress. Each figure represents the ratio of the total time the flow spends in each quadrant at different axial positions and normalized distances from the suction side of the high pressure compressor blade.

Figure 6.15 shows the quadrant analysis results of the position $x/x_{lax} = 0.1$. At the first boundary layer section, Q2 and Q4 are dominant throughout all points normal to the wall, with Q4 being the most dominant contributor. High values of Q2 highlight ejections that likely involve low-momentum fluid being pushed away from the boundary layer into the free-stream. High values of Q4 highlight sweep events where high-momentum fluid from the free-stream is brought towards the wall. Q3 has minimal representation, suggesting infrequent inward movements of fluid towards the wall at this pre-separation point. Q1, representing the outward motion, is almost negligible. The reason for Q4 being the highest contributor is the high receptivity of the boundary layer to free-stream turbulence intensity at the early stage of the flow on the suction side.

When moving to the second boundary layer cut at $x/x_{lax} = 0.35$ (cf. Fig. 6.16), there is a slight increase of Q3 event compared to the pre-separation (pre-transitional) region, reflecting enhanced turbulent mixing as the separation process begins. The increase of Q1 potentially indicates the outward transport as the separated flow starts to interact with the free-stream. Q2 is the most prominent event across a range of wall-normal distances. These ejections indicate a significant presence of upward-moving turbulent bursts, which are characteristic of separated flow regions where low momentum fluid is detached from the wall. Q4 remains prominent near the wall, suggesting ongoing high-momentum sweep events that contribute to maintaining some level of momentum near the wall despite separation.

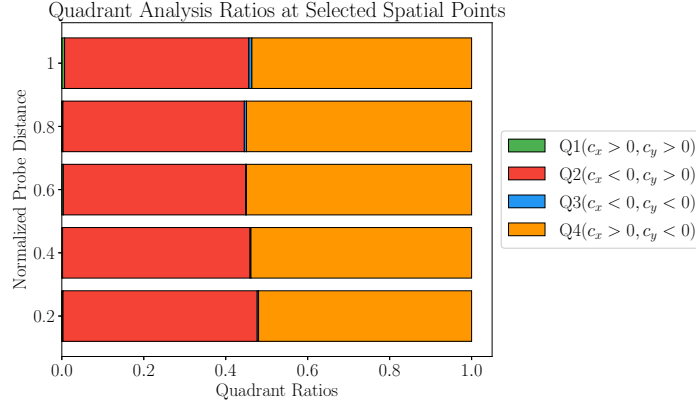


Figure 6.15: Quadrant analysis based on iLES and at mid-span at boundary layer section $x/x_{lax} = 10\%$

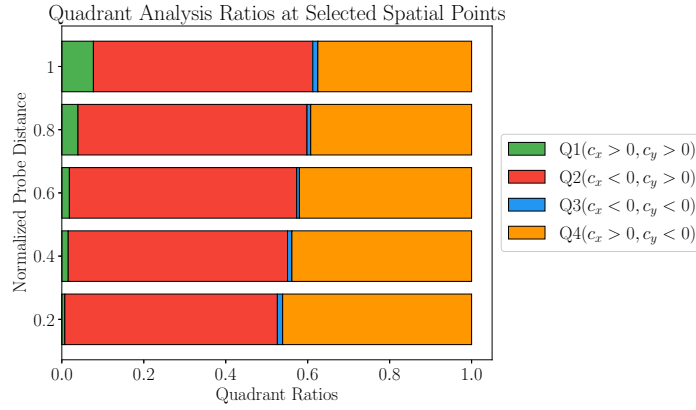


Figure 6.16: Quadrant analysis based on iLES and in mid-span at boundary layer section $x/x_{lax} = 35\%$

At the last boundary layer section ($x/x_{lax} = 0.8$) shown in Fig. 6.17, where the flow is reattached and fully turbulent, the order of the quadrant ratios changes slightly, compared to the previous boundary layer locations. Q1 and Q3 events are more prominent when moving away from the wall. On the other hand, Q2 and Q4 dominate the region near the wall, highlighting the recovering boundary layer, where ejections (Q2) help in mixing and redistributing momentum, and at the same time, the continued influence of sweeps (Q4) in bringing high-momentum fluid towards the surface helps in boundary layer stabilization and thickening of the boundary layer in post-reattachment region. The quadrant analysis results highlight the complexity and diversity of each quadrant event to the overall Reynolds shear stress at different boundary layer states. The quadrant analysis results help us to understand the dynamics of the boundary layer and the interaction of FST and the boundary layer. The TKE distribution identifies the regions of strongest turbulence activity. The quadrant analysis then reveals that in these regions, ejections and sweeps are the dominant contributors, consistent with strong shear-layer turbulence production

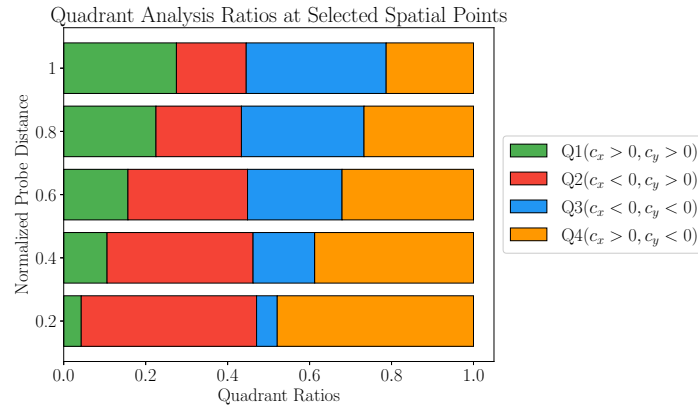


Figure 6.17: Quadrant analysis based on iLES and in mid-span at boundary layer section $x/x_{lax} = 80\%$

6.3 Influence of the free-stream turbulence on the flow at mid-span

In the previous sections, the mid-span flow of the high-pressure compressor cascade was analyzed using iLES results with respect to flow losses and turbulence behavior, and these results were compared with best-practice RANS results. In this section, the influence of different free-stream turbulence levels on the mid-span flow of the high-pressure compressor cascade is examined for the operating point $Re_{in} = 492,000$, $M_{1,th} = 0.4$. The analysis focuses on critical design parameters, the isentropic Mach number distribution over the blade and the total pressure loss in the wake. For this purpose, three iLES cases with varying free-stream turbulence intensities at the inlet were conducted, denoted as iLES-Clean, iLES-2.5% and iLES-6%. No turbulent length scale variation was performed in these simulations.

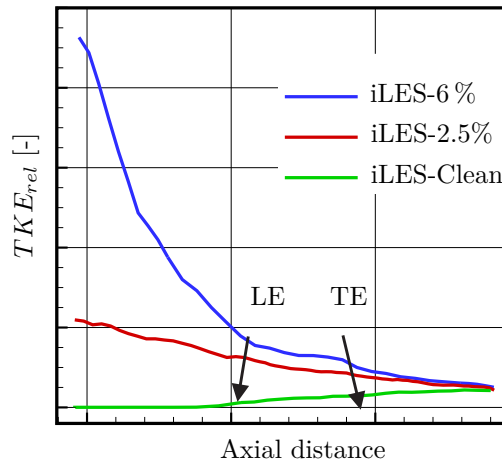


Figure 6.18: Normalized turbulent decay along a streamline at mid-pitch, mid-span by iLES for three free-stream turbulence levels

As already stated in Sec. 5.3, due to the limitations of prescribing higher turbulent length scales than the dimensions of the three-dimensional configurations, the largest turbulent length scale permissible for the given configuration was used. For this study, utilizing larger turbulent length scales would have required more than one pitch (as a simulation model), which was computationally prohibitive. As a result, the investigation is restricted to a single passage. The turbulent decay along the mid-pitch and mid-span throughout the domain for the three free-stream turbulence intensities is shown in Fig. 6.18. Before delving deeper into the results, the following key aspects need to be highlighted

1. Turbulent Length Scale

The high decay rate observed in the iLES-6 % free-stream turbulence case is directly linked to the relatively small turbulent length scale prescribed at the inlet, approximately 4 % of the chord length. This steep decay is further promoted by the higher Reynolds stresses associated with the increased turbulence intensity in the iLES-6 % case. It should be noted again that the same turbulent length scale is prescribed for all iLES cases, and the turbulence level is controlled via the turbulence intensity, e.g., the Reynolds stress tensor. Due to the aforementioned restriction, the turbulence intensity of the iLES-6 % case at the leading edge, reduced to a value around 3 %.

2. Simulation Convergence

The iLES-Clean and iLES-6 % cases were run for fewer through-flows than the iLES-2.5% case. While the simulation time for these two cases may not fully resolve all turbulent effects, it is considered sufficient to provide meaningful insights into the influence of free-stream turbulence on the compressor cascade flow.

3. Side-wall Boundary Layer Representation

The prescribed side-wall boundary layer is identical across all cases and corresponds to a turbulent boundary layer derived from and matched to experimental data. An interesting extension of this work would involve examining the impact of prescribing a laminar side-wall boundary layer, which could influence surface and wake characteristics.

Despite these factors, the study establishes a reliable foundation for exploring the effects of free-stream turbulence on mid-span flow and turbulence behavior of the compressor cascade. Future investigations could refine these findings by addressing the influence of varying turbulent length scales, extended simulation times, and different prescription of the boundary layer.

The results of Fig. 6.18 and subsequent observations show that the turbulence intensity (or TKE) for iLES-2.5% and iLES-6 % cases, continuously decays throughout the domain due to viscous dissipation, as expected for higher free-stream turbulence levels. The iLES-Clean case, starts with very low turbulence intensity at the inlet, but exhibits an increase in TKE shortly after entering the blade passage. This growth is attributed to turbulence production mechanisms within the blade passage. The iLES-Clean case shows, interestingly, very similar value as the other two cases, in the wake area. All cases, iLES-Clean, iLES-2.5%, and iLES-6 %, reach nearly identical TKE levels in the wake region, with only the iLES-6 % case maintaining slightly higher values. This convergence highlights that, while initial free-stream turbulence influences TKE development in the passage, the wake flow dynamics tend to homogenize these differences. This behavior is strongly related to

the steep turbulent decay in the free-stream flow. In the subsequent figures, the similarities and discrepancies regarding the development of turbulence between these cases will be discussed.

The results presented in Fig. 6.19 provide critical insights into the effects of free-stream turbulence intensity on the isentropic Mach number distribution and boundary layer behavior over the blade. There are three major regions, which are affected by the change of the free-stream turbulence.

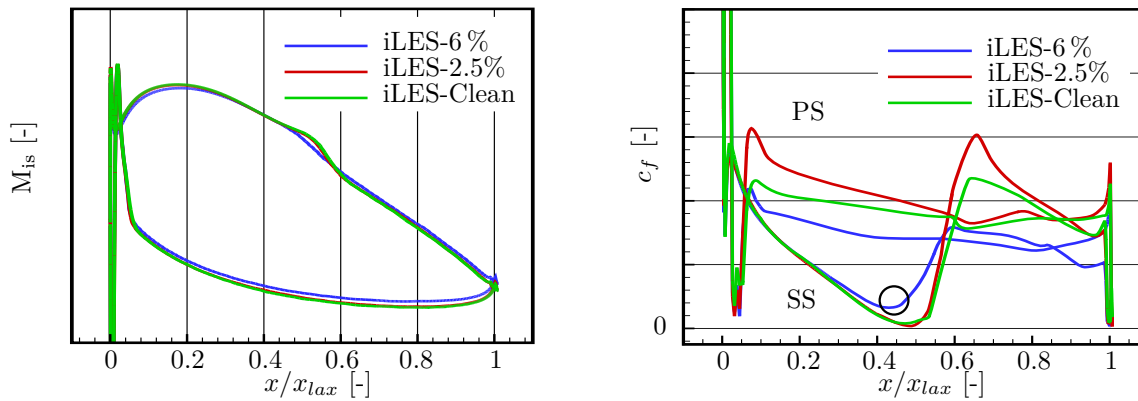


Figure 6.19: Normalized isentropic Mach number distribution and skin-friction coefficient over the high-pressure compressor blade by iLES for three free-stream turbulence levels

1. On the first half of the suction side, the iLES-6 % case, exhibits a distinctly different isentropic Mach number distribution compared to the other cases (lower suction side and higher pressure side Mach number distribution). This discrepancy can not be attributed to variations in the inflow angle, as all cases share an identical inflow angle at the leading edge. The observed differences arise purely due to variations in the free-stream turbulence intensity. Higher free-stream turbulence alters the boundary layer development early on the suction side, influencing the local pressure distribution.
2. The next major difference is observed at the middle of the suction side ($x/x_{lax} = 0.5$), which is related to the transition mechanism. By increasing the free-stream turbulence level from the experimental values (2.5%) to 6 %, the transition mode and the isentropic Mach number distribution change accordingly. By increasing the free-stream turbulence, the pressure jump typically associated with the separation bubble, disappears. At higher free-stream turbulence levels (6 %), the boundary layer transitions to turbulence earlier, preventing the flow from separation. Increased free-stream turbulence enhances low-frequency streamwise streaks (Klebanoff modes), which promote the breakdown of Kelvin-Helmholtz structures. This leads to the suppression of separation and reduces the length of the separation bubble. These findings align with Zaki et al. (2010), who demonstrated that higher free-stream turbulence intensities increase the energy carried by Klebanoff modes, accelerating transition and modifying boundary layer behavior in compressor cascade flow.

3. On the pressure side, there is a moderate pressure gradient, and turbulence has a smaller impact compared to the suction side. The higher turbulent intensities (iLES-6 %) slightly enhance the boundary layer stability and reduce the boundary layer thickness. As a consequence, there is a slightly different pressure recovery.

In general, there is a reduction in blade loading for the iLES 6 % case, compared to the other two cases. Figure 6.19 illustrates the skin-friction development over the profile for all three cases. The presence of higher free-stream turbulence significantly influences the transition onset, which is defined by the minimum value of the skin-friction coefficient. The transition onset is shifted approximately 10 % of the axial chord upstream for the iLES-6 % case (compared to the other two cases). The increased turbulence intensity suppresses the separation bubble and triggers an earlier transition to turbulence. The resulting turbulent boundary layer exhibits higher skin-friction (shown as black circle in Fig. 6.19), which can slightly reduce the peak isentropic Mach number compared to a laminar or separated flow. Between the iLES-Clean case and the iLES-2.5% case, there is a marginal difference in the skin-friction coefficient, in terms of transition onset (upstream axial shift of $\approx 1.5\%$). In general, the iLES-2.5% case predicts higher skin-friction values than iLES-Clean on both suction and pressure surfaces. On the pressure side, the iLES-6 % case shows a small axial offset relative to the other two cases, in terms of skin-friction distribution.

Figure 6.20 illustrates the influence of free-stream turbulence on the total pressure losses in the wake planes. At the first axial wake plane, the iLES-Clean case predicts lower wake flow losses compared to the other two cases. This difference is particularly noticeable at the peak flow loss values. At this axial position, there is no significant difference between the iLES-2.5% and iLES-6 %.

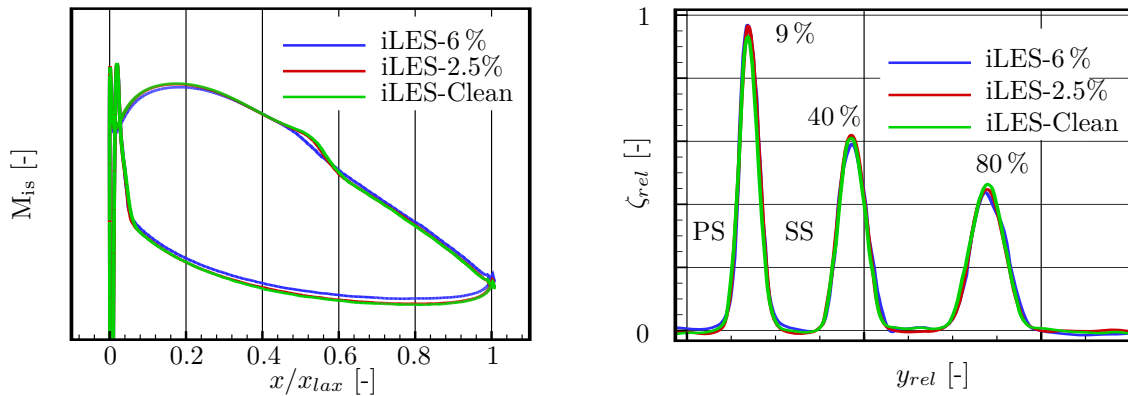


Figure 6.20: Normalized isentropic Mach number distribution and normalized total pressure loss in wake by iLES for three free-stream turbulence levels

Further downstream, the differences become more prominent, while at the last axial wake plane, the iLES-Clean case exhibits the highest peak flow loss values. In general, the iLES-Clean case shows a slower decay of flow losses compared to the other two cases, which can be attributed to the slower mixing of flow losses in the wake. As the flow progresses further downstream, particularly at the second and last axial planes, the iLES-6 % case demonstrates the lowest wake flow losses. The iLES-2.5% case behaves similarly to the iLES-6 % case, except at the wake plane 40 % downstream of the trailing edge, where a slight deviation is observed. The distinct behavior of the iLES-6 % case is primarily

associated with the different boundary layer characteristics on the suction side of the blade. Similar trends in the wake region have been observed experimentally by Lang (2024). Although the results of Lang (2024) were obtained at a different operating point for the same geometry, the findings align well with the trends reported in his study.

Figure 6.21 presents the contours of streamwise vorticity for all three cases, highlighting the influence of free-stream turbulence in the passage and wake regions. In the iLES-Clean case (left contour), no streamwise vorticity is observed in the free-stream due to the absence of inflow turbulence. However, vorticities begin to develop in the rear part of the suction side and over the pressure side, which are associated with the separation bubbles in these regions. In contrast, for the iLES-2.5% and iLES-6 % cases, streamwise vorticity induced by the free-stream turbulence is clearly visible from the leading edge plane. Moreover, the elevated free-stream turbulence significantly increases both the distribution and density of the streamwise vorticity throughout the domain.

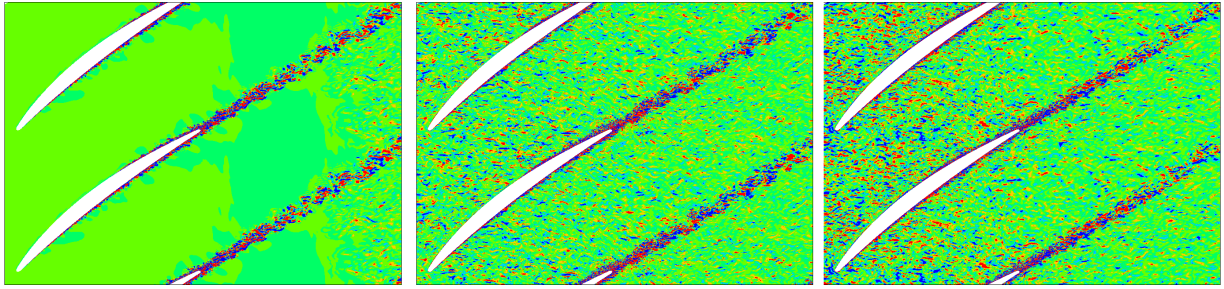


Figure 6.21: Streamwise vorticity contours by iLES for three free-stream turbulence levels. Left: iLES-Clean, Middle: iLES-2.5%, Right: iLES-6 %

Figure 6.22 shows the normalized TKE distribution near the leading edge for all three cases. For the iLES-2.5% and iLES-6 % cases, there is a region of high TKE on the pressure side near the leading edge, with minor differences between them (iLES-2.5% shows slightly larger separation bubble than iLES-6 %). These regions are associated with the separation bubble on the pressure side, as previously discussed. The iLES-Clean case exhibits significantly lower TKE values compared to the other two cases.

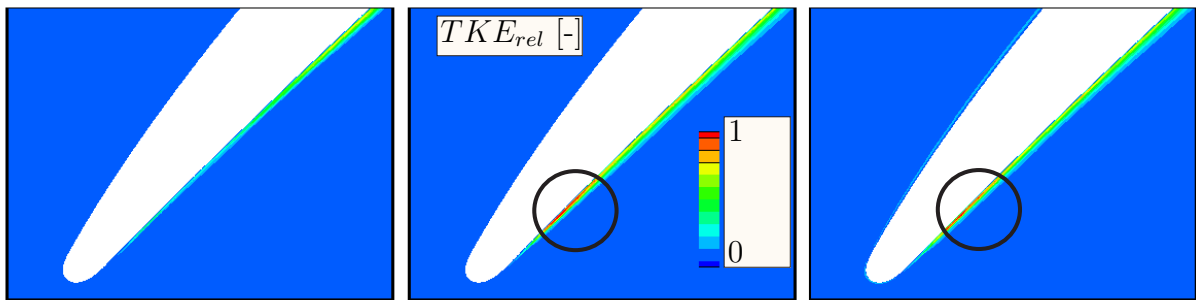


Figure 6.22: Zoomed view of the normalized TKE contours by iLES for three free-stream turbulence levels. Left: iLES-Clean, Middle: iLES-2.5%, Right: iLES-6 %

On the suction side (Fig. 6.23), the differences between the iLES-2.5% and iLES-6 % are marginal, with the size of the separation bubble playing a critical role. The iLES-6 % case shows slightly earlier rise of TKE and a more flat distribution. This behavior is

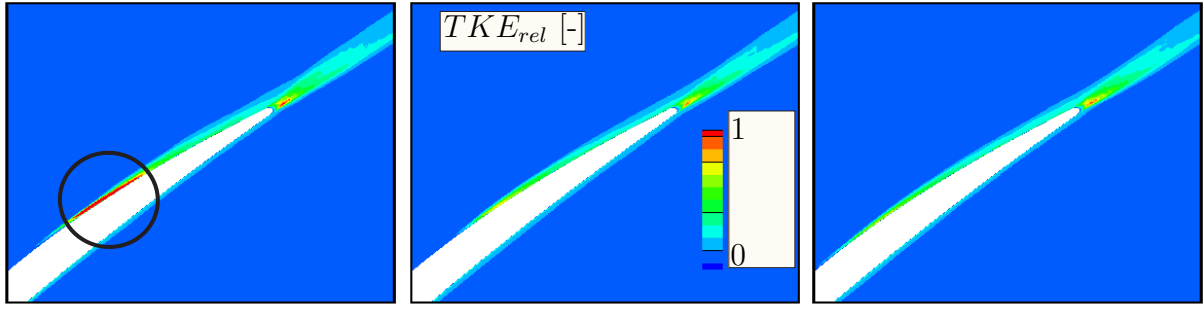


Figure 6.23: Zoomed view of the normalized TKE contours by iLES for three free-stream turbulence levels. Left: iLES-Clean, Middle: iLES-2.5%, Right: iLES-6 %

related to the influence of the elevated free-stream turbulence on the TKE evolution in the boundary layer. The iLES-Clean case reveals the biggest difference between the three cases, with the highest TKE values in the separation bubble and near the wake area. Because of the missing free-stream turbulence, in the iLES-Clean case, the TKE values are more pronounced in these regions. As the flow moves towards the trailing edge and wake area, regions of high TKE values are identified, again linked to the more pronounced separation bubble. iLES-Clean case is solely controlled by the flow behavior itself and not by the free-stream turbulence. The shear stress contour plots (Fig. 6.24) highlight regions

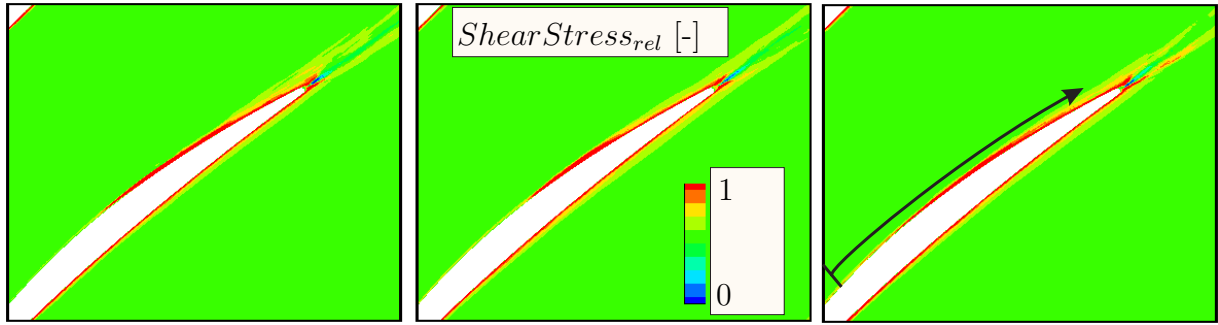


Figure 6.24: Zoomed view of shear stress contours by iLES for three free-stream turbulence levels. Left: iLES-Clean, Middle: iLES-2.5%, Right: iLES-6 %

where the flow transitions to a fully turbulent state. Overall, the shear stress distributions do not show significant differences between the cases, offering limited additional insights. However, it can be observed that in the iLES-6 % case, shear stresses begin to increase earlier compared to the other two cases. This behavior is attributed to the elevated free-stream turbulence, which penetrates the boundary layer, suppresses the separation bubble, and accelerates the transition to turbulence, as previously discussed.

Figure 6.25 illustrates the evolution of TKE, along boundary layer cuts, over the suction side for all three cases. In the first half of the suction side, all three cases exhibit a continuous increase in the maximum normalized TKE as the flow moves downstream from $x/x_{lax} = 10\%$ to $x/x_{lax} = 40\%$. Additionally, the maximum TKE values shift progressively further away from the surface in the downstream direction. This behavior, as previously shown in the LPT cascade MTU-T161 case, is attributed to the thickening of the boundary layer and the corresponding shear layer moving away from the wall.

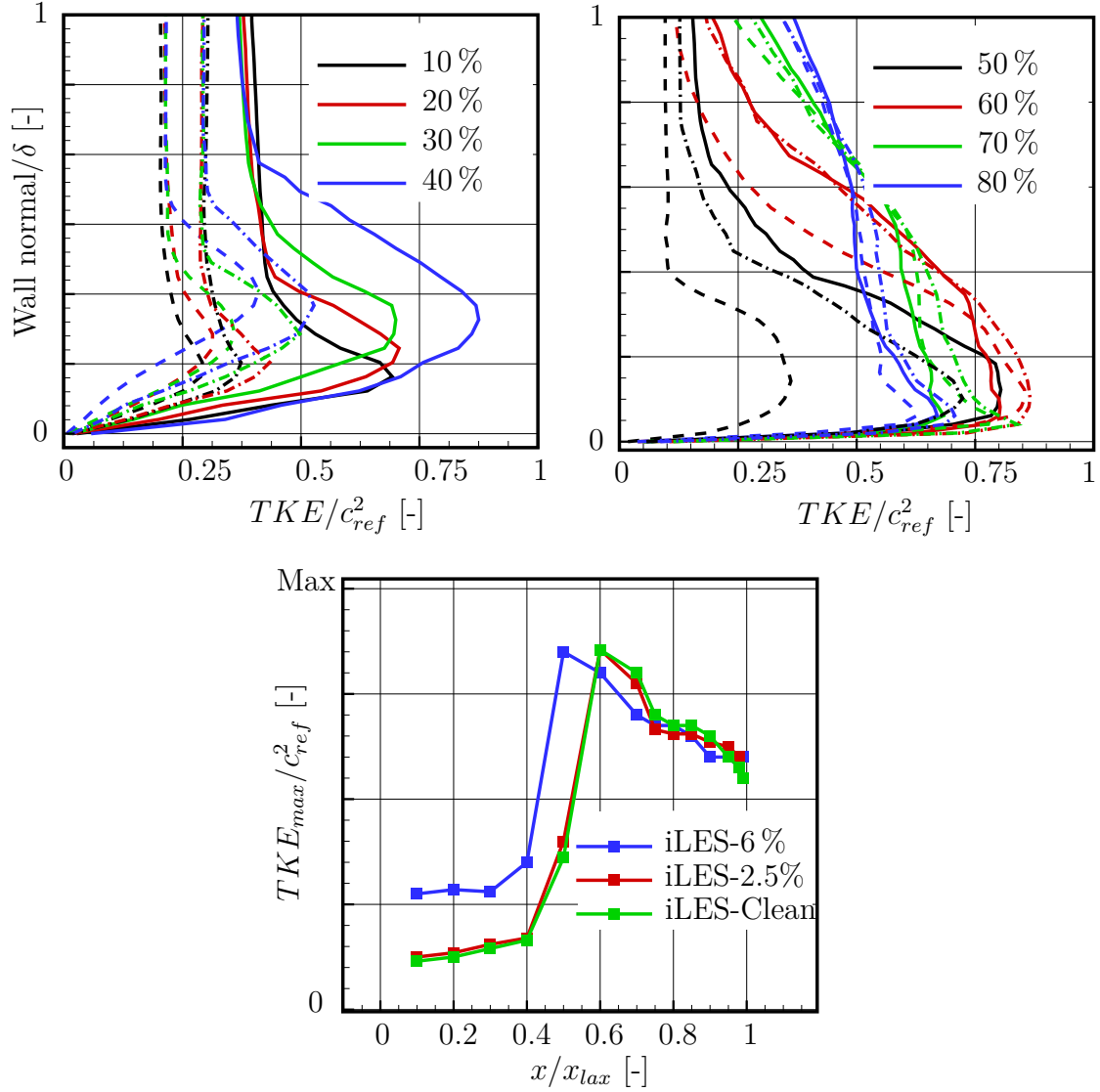


Figure 6.25: Top row: Evolution of normalized TKE over the suction side by iLES for three free-stream turbulence levels. Solid: iLES-6 %, Dashed-dotted: iLES-2.5%, Dashed: iLES-Clean. Bottom row: Evolution of TKE_{max} over the suction side by iLES for three free-stream turbulence levels

In the first half of the suction side flow, iLES-6 % case shows consistently higher TKE values followed by iLES-2.5% and iLES-Clean. The differences in elevated TKE values are most noticeable at $x/x_{lax} = 40\%$. Here, the ratio of the maximum TKE values between the iLES-6 % and iLES-2.5% cases is approximately 1.75, while the ratio between the iLES-2.5% and iLES-Clean case is about 1.3. By comparing all three cases, it becomes evident that an increase in free-stream turbulence intensity leads to a corresponding increase in the maximum TKE values.

In the second half of the suction side, notable differences arise compared to the first half. Overall, the discrepancies between the three cases diminish in the second half with respect to TKE behavior. However, at $x/x_{lax} = 50\%$, minor differences in TKE values remain observable. As the flow transitions into a fully turbulent state, all three cases show rather

similar TKE peak values. At the second half of the suction side flow, it is observed that the iLES-Clean and iLES-2.5% cases exhibit even slightly higher TKE values than the iLES-6% case, which differs from the behavior seen in the first half of the suction side flow. Figure 6.25 highlights the evolution of the $TKE_{\max}$ position along the suction side for all three cases. A similar analysis was previously conducted for the low-pressure turbine cascade MTU-T161. The results here confirm the earlier findings. The iLES-Clean and iLES-2.5% cases exhibit a very similar trend in the development of $TKE_{\max}$, with only marginal differences in the second half of the suction side. In contrast, the iLES-6% case reveals that the $TKE_{\max}$ peak shifts closer to the leading edge. In this case, the $TKE_{\max}$ values are already elevated near the leading edge compared to the other two cases.

6.4 Challenges of RANS in predicting compressor flows

In the previous sections, the potential and validity of iLES in predicting important design variables, such as blade pressure distribution and wake flow losses, have been demonstrated, whereas these variables are not predicted with the desired accuracy by conventional RANS methods. For this purpose, the iLES results were compared with best-practice linear eddy viscosity RANS results. Furthermore, the iLES results were analyzed with respect to turbulence behavior at mid-span. In this section, some shortcomings of the RANS methods, as highlighted in this chapter, will be reviewed and summarized.

Boussinesq assumption

As already stated in Sec. 5.4, the suitability of the two-equation linear eddy viscosity turbulence modeling framework can be assessed by quantifying the degree of non-linearity in the stress-strain relationship. Figure 6.26 illustrates the alignment ratio (C_{as}) for the RANS simulation of the high-pressure compressor cascade. The whole flow is dominated by ratios < 0.75 , demonstrating the difficulties of the RANS turbulence model, used in this chapter, to represent the correct alignment of the stress-strain ratio. In the region of separation and shear layer, the alignment ratio reduces even further.



Figure 6.26: Alignment of the shear rate tensor to the anisotropy tensor based on RANS result

As previously outlined by Tucker (2014), the majority of linear eddy viscosity-based RANS models are calibrated for flows that are predominantly characterized by simple shear

strains. In case of dominant compressive or extensive strains, they can result in significant inaccuracies in fundamental RANS models. The elevated compressive strain in stagnation regions results in RANS models significantly overestimating turbulence production. This over-prediction results in the suppression of leading edge separation on blades, as demonstrated by Medic et al. (2002), and can also have significant downstream impacts. Therefore, turbulence production corrections, such as the Kato et al. (1993) correction or the TKE production limiter in the SST model, are required. However, these corrections have unintended consequences elsewhere in the flow and have resulted in other formulations over the years.

Turbulence Anisotropy

The turbulence anisotropy, already discussed before, is a challenging task for conventional RANS turbulence models. Especially, the misalignment angle between Reynolds stress and shear strain (cf. Fig. 6.26), a major drawback of the linear eddy viscosity turbulence models, can explain why it is hard for linear eddy viscosity turbulence models to correctly predict specific flow features like flow separation. Figure 6.6 demonstrates that the RANS turbulence model used in this chapter predicts turbulence anisotropy quite differently compared to iLES. The iLES results show that all turbulence states (1C, 2C and 3C) are present at different regions of the compressor cascade flow. The complexity of the turbulence states highlights the drawbacks of conventional linear eddy viscosity RANS turbulence models.

TKE evolution over suction side

As shown in Sec. 6.1.2, the RANS model used in this chapter overestimates the TKE values in the transitional and pre-transitional regions over the suction side (compared to the iLES result). As discussed previously, RANS models use the Boussinesq hypothesis to relate Reynolds stresses to mean flow gradients. During transition, turbulence production is highly non-linear, anisotropic, and intermittent, which the linear eddy viscosity models are not able to capture correctly. It is important to note that these assumptions can result in an overestimation of turbulence production. This phenomenon is commonly regulated via additional modifications, as previously mentioned. These modifications have been shown to artificially reduce the TKE growth. In physical terms, the boundary layer may remain laminar for a greater duration, and TKE remains lower (than iLES) in transitional regions. The production term, as modelled, assumes fully developed turbulence and fails to account for the contribution from early instabilities (Klebanoff modes). The dissipation rate in RANS models is frequently overestimated in transitional regions, thereby diminishing the growth of TKE. The modelling of turbulent transport is inadequate, resulting in an insufficient spreading of turbulence into transitional regions. As previously outlined, the aforementioned factors result in a reduced net growth of TKE in comparison to that observed in high-fidelity simulations.

The RANS turbulence models, used in this work, aim to fix some of the shortcomings mentioned before, but the results and comparisons show that these models are unable to overcome these issues properly. In general, the correct prediction of transitional region of the flow is, despite the advancement of transition models over the last decades, a major challenge for RANS models. The correct prediction of flow separation and consequently the reattachment, remains another challenge of the RANS turbulence models, as already shown in the current chapter.

6.5 A note on computational cost

In this section, the computational costs of the high-fidelity simulation of the high-pressure compressor cascade are compared with those of the RANS simulation, as computational efficiency is considered a key factor in assessing the applicability of high-fidelity methods. For this purpose, the computational cost of the iLES 5th-order simulation is evaluated, as this mesh was found during the validation phase to be sufficiently fine to capture the necessary physics. The iLES calculation was performed on a CPU system using Intel Skylake processors at the high-performance computing (HPC) cluster of RWTH Aachen University. It required a wall clock time of 1,440 hours on 2,400 CPUs for 20 convective times, amounting to approximately 3,500,000 core-hours. In contrast, the RANS calculation is approximately 800 times faster, requiring only 4,000 core-hours. This comparison highlights the efficiency of RANS models and explains why they remain the primary tool in industrial design workflows. However, as demonstrated in this chapter, RANS models still struggle to accurately predict blade loading and flow losses at an off-design point. In the case of the high-pressure compressor cascade, which operates at a higher Reynolds number than the low-pressure turbine cascade, the computational cost gap between RANS and high-fidelity methods becomes even more significant (cf. Sec. 5.5).

7 Summary and Outlook

In this thesis, the flow over a highly loaded low-pressure turbine (LPT) cascade and a high-pressure compressor (HPC) cascade is investigated using large eddy simulation (LES) and validated against experimental data. Both configurations were simulated as three-dimensional models, including side-walls, but the focus remains on the mid-span flow. The operating conditions in both cases correspond to off-design points, highlighting the role of transition and separation. A mesh sensitivity study was conducted, ensuring grid convergence for key parameters such as the isentropic Mach number distribution and total pressure loss in the wake.

The LES results of the LPT cascade are compared against best-practice RANS simulations in terms of surface values (isentropic Mach number and skin-friction), total pressure loss, and turbulent kinetic energy (TKE), demonstrating the importance of accurate transition prediction for flow loss estimation. The most significant difference between LES and RANS lies in the prediction of the separation bubble. RANS fails to capture the suction-side separation bubble as observed in both LES and experiments, leading to inaccurate predictions of blade loading. RANS underpredicts the separation bubble on the suction side while overestimating it on the pressure side. Examining the TKE evolution along the suction side reveals that in the acceleration region, RANS predicts up to four times lower TKE than LES, reflecting the modeling difficulties of RANS models in transitional flows. However, after separation, where the flow becomes fully turbulent, the discrepancy between the two models becomes considerably smaller. At the wake plane, RANS predicts an integral total pressure loss that is 32 % lower than LES, and turbulence production at the wake is underestimated by a factor of 2.5. Further analysis of anisotropy confirms that RANS struggles to reproduce the full turbulence state, particularly in the separated shear layer, due to the limitations of linear eddy viscosity models.

To better assess flow losses, entropy generation is analyzed. LES reveals that over 90 % of the total entropy is generated along the separated shear layer, with peak entropy locations differing from those predicted by RANS. This discrepancy originates from the different separation bubble predictions and the resulting turbulence characteristics. Among the contributing terms, turbulence production emerges as the dominant factor contributing to entropy generation. The LES results were further analyzed using spectral, modal, and quadrant analyses. Spectral analysis identifies low-frequency fluctuations before separation, with a shift toward higher frequencies during transition and reattachment. The modal analysis, based on the spectral proper orthogonal decomposition (SPOD) method, reveals that the dominant mode (Mode 0) captures the large-scale coherent structures such as separation bubbles and wake vortices, while Mode 1 represents secondary vortices and detached shear layers. The SPOD analysis effectively links flow phenomena to specific modes, frequencies, and spatial regions, providing deeper insight into turbulence dynamics. The quadrant analysis of Reynolds shear stresses shows that ejection (Q2) and sweep (Q4)

events dominate before separation, contributing to momentum redistribution, while Q1 and Q3 interactions increase downstream, supporting reattachment and boundary layer recovery.

A parametric study was conducted to investigate the impact of free-stream turbulence (FST) on boundary layer development and wake flow losses. Three different turbulence intensities were tested (LES-Clean, LES-4 % and LES-8 %). The results show that higher turbulence intensities reduce separation bubble size by a factor of 1.2 and shift the transition onset upstream. The integral flow losses are 2.8 times higher in LES-Clean than in LES-8 %, mainly due to open separation bubble on the suction side surface. Increased free-stream turbulence enhances boundary layer entrainment, alters TKE distribution, and suppresses separation.

Similar to the LPT case, the HPC cascade results are compared against best-practice RANS simulations. The most significant discrepancy lies, again, in the prediction of the separation bubble. RANS fails to capture the suction-side separation bubble observed in both LES and experiments. Examining TKE evolution, RANS overpredicts the peak TKE by up to seven times in the first half of the suction side, preventing bubble formation. However, in the second half, RANS underpredicts TKE after reattachment, a trend similar to that observed in the LPT case over the entire suction side surface. At the wake plane, RANS predicts 55 % higher integral total pressure losses than LES. The LES results agree very well with the measurements.

The entropy analysis of the high-pressure compressor cascade using both LES and RANS identifies regions of significant entropy generation and associated flow losses. LES predicts high entropy generation due to turbulence production in the suction and pressure-side boundary layers as well as the wake. On the suction side, entropy generation begins in the mid-blade region, marking the laminar-to-turbulent transition, while on the pressure side, it starts from leading edge. In wake region, two distinct entropy peaks are observed, corresponding to contributions from both boundary layers and vortex shedding downstream of the trailing edge, in line with the TKE distribution. In contrast, RANS shows turbulence-driven entropy generation in the boundary layer, starting at the leading edge and extending over the entire suction side, creating a thicker region of entropy generation. In wake region, RANS captures two wake peaks similar to LES, however its magnitude and resolution are less pronounced, supporting the overestimated wake flow losses. Spectral analysis reveals that near the leading edge, velocity fluctuations are broadly distributed with dominant low-frequency content, driven by free-stream turbulence. Further downstream, energy concentrates in higher frequencies and shifts toward the wall. As transition and separation occur, energy moves to higher frequencies and becomes more spatially dispersed, with similar trends observed on both surface sides. These spectral changes indicate the onset of transition and reattachment, with fully turbulent states displaying a more uniform high-frequency energy distribution.

The quadrant analysis of the Reynolds stress tensor along the mid-span of the high-pressure compressor cascade reveals distinct turbulence contributions at various axial positions. At early stages of the boundary layer, the boundary layer is dominated by Q2 (ejections) and Q4 (sweeps), with Q4 being particularly pronounced due to the high receptivity to free-stream turbulence. Moving downstream, there is a modest increase in Q1 and Q3 events, indicating enhanced turbulent mixing as the separation process begins, while Q2 remains the most prominent near the wall. In the reattached and fully turbulent region, Q1 and

Q3 become more significant away from the wall, whereas near-wall dynamics continue to be governed by Q2 and Q4, which facilitate momentum redistribution and boundary layer stabilization. These findings highlight the intricate interplay of different turbulence events in shaping the overall Reynolds shear stress across various boundary layer states.

Similar to the LPT case, the flow over the HPC cascade was analyzed with varying free-stream turbulence intensities (LES-Clean, LES-2.5%, and LES-6%), revealing significant effects on flow behavior. The results of LES-Clean and LES-2.5% are nearly identical in terms of the isentropic Mach number distribution over the blade surface. In contrast, for LES-6%, the separation bubble on the suction side is suppressed due to the increased free-stream turbulence. However, the wake flow loss results show only marginal differences, as the turbulent mixing in the wake and the flow behavior over the second half of the suction side are similar across all cases. Examining the TKE evolution over the suction side, LES-6% produces the highest TKE in the first half of the suction side, but the differences diminish in the second half.

The computational cost analysis highlights the trade-off between accuracy and efficiency. The LES simulation of the HPC cascade required 3.5 million core-hours on 2,400 CPUs, while the corresponding RANS simulation was 800 times faster, consuming only 4,000 core-hours. Similarly, for the LPT cascade, LES required 387,000 core-hours compared to 3,800 core-hours for RANS. These results illustrate why RANS remains the industry standard, offering up to 800 times speed advantage over LES. However, the inability of RANS to accurately capture transition and separation highlights the growing necessity of LES for high-fidelity turbomachinery simulations.

This work highlights the limitations of RANS models and provides valuable high-fidelity data for improving future turbulence and transition models, particularly for enhancing CFD workflows in industry. In particular, the high-resolution dataset generated in this work could be used to develop and validate machine-learning-based turbulence models, offering improved predictions of laminar separation bubbles and transition processes. Furthermore, future research should explore secondary flow effects, particularly corner separations and their impact on overall flow losses, as these factors are critical for improving industrial CFD accuracy. Further studies should also focus on extending these high-fidelity simulations to more complex configurations, such as multi-stage compressors and turbines, to better understand the influence of blade interactions on flow loss mechanisms. Moreover, incorporating wall-resolved LES for higher Reynolds number cases could help improve transition modeling at industrially relevant conditions. Another promising direction is the refinement of hybrid RANS-LES models (or wall-modeled LES), which aim to balance accuracy and computational cost. By leveraging the insights from this work, improved turbulence closures could be developed that selectively apply LES-based corrections in critical regions such as transition zones and wakes, while maintaining RANS-like efficiency elsewhere.

In addition, future research should examine how free-stream turbulence characteristics, including turbulent length scales, influence transition and separation under more realistic inflow conditions. Experimental validation of these findings in large-scale test rigs would further enhance the applicability of LES-generated data to industrial design workflows. Finally, the combination of high-fidelity simulations with data-driven methods, such as machine learning-assisted turbulence modeling and reduced-order modeling techniques,

could enable real-time predictions of aerodynamic losses in turbomachinery. This integration would significantly accelerate the design cycle while maintaining the accuracy of high-fidelity approaches.

Nomenclature

Latin letters

Symbol	Unit	Description
a_{ij}	—	Anisotropy tensor
d	m	Diameter
e	m^2/s^2	specific inner energy
p	Pa	Pressure
c	$\frac{\text{m}}{\text{s}}$	Velocity
l_{ax}	m	Axial length
k	m^2/s^2	Turbulent kinetic energy
l	m	Length/Chord length
M	—	Mach number
T	K	Temperature
P	m^2/s^3	Turbulence Production
Re	—	Reynolds number
Re_{ind}	—	Indifference-Reynoldszahl
$Re_{\theta t}$	—	Critical-Reynoldszahl
G_s	$\frac{\text{J s}}{\text{K}}$	Overall entropy generation
P_R	$\frac{\text{J s}}{\text{K}}$	Entropy generation related to turbulence production
V_m	$\frac{\text{J s}}{\text{K}}$	Entropy generation related to viscous dissipation
A_{TKE}	$\frac{\text{J s}}{\text{K}}$	Entropy generation related to turbulence advection
Zw	—	Zweifel number
x	m	Position
c_f	—	Skin-friction coefficient
y	m	Wall-normal distance
S	1/s	Mean strain rate
H	m^2/s^2	Total enthalpy
c_p	$\text{m}^2/\text{s}^2\text{K}$	Isobar heat capacity
c_v	$\text{m}^2/\text{s}^2\text{K}$	Isochoric heat capacity
c_f	—	Skin friction coefficient
C_p	—	Pressure coefficient
s	$\frac{\text{J}}{\text{K}}$	Entropy
R	$\text{m}^2/\text{s}^2\text{K}$	Specific gas constant
t	—	Total

Symbol	Unit	Description
E	—	Energy spectrum function
f	Hz	Frequency
p	—	Polynomial approximation order of the solution
n	—	Number
TI	—	Turbulence intensity
CI	—	Confidence interval
s	—	Variance
S	1/s	Shear rate tensor
Q	—	Quadrant events
t_c	—	Convective time unit
i	°	Incidence angle
q_i	J/sm ²	Heat flux vector
y_+	—	Dimensionless wall normal distance
x_+	—	Dimensionless wall streamwise distance
z_+	—	Dimensionless wall spanwise distance
l_Λ	m	Integral turbulent length scale

Greek letters

Symbol	Unit	Description
ρ	$\frac{\text{kg}}{\text{m}^3}$	Density
γ	—	Intermittency
γ	—	Isentropic exponent
η	$\frac{\text{kg s}}{\text{m}}$	Dynamic viscosity
η	m	Kolmogorov length scale
τ_η	s	Kolmogorov time scale
τ_{ij}	kg/ms ²	Stress tensor
$\tau_{v,ij}$	kg/ms ²	Viscous stress tensor
ζ	—	Total pressure loss coefficient
β_1	°	Inflow angle
β_2	°	Outflow angle
τ_w	kg/ms ²	Wall shear stress
δ	m	Boundary layer thickness
ϵ	m ² /s ³	Specific dissipation
κ	m	Wave number
ν	m ² /s	Kinematic viscosity
ω	1/s	Specific dissipation rate
λ	1/s	Eigenvalue, POD mode
σ	—	Standard deviation
μ_t	$\frac{\text{kg s}}{\text{m}}$	Turbulent viscosity

Symbol	Unit	Description
λ_θ	—	Acceleration parameter
ω	1/s	Vorticity
θ	m	Boundary layer thickness
μ	$\frac{\text{kg s}}{\text{m}}$	Molecular viscosity
Ω	1/s	Absolute vorticity

Subscripts

Subscript		Description
<i>is</i>	—	Isentropic
<i>th</i>	—	Theoretical
<i>out</i>	—	Outflow
<i>in</i>	—	Inflow
<i>t</i>	—	Total
<i>rel</i>	—	Relative
<i>ref</i>	—	Reference
<i>max</i>	—	Maximum
'	—	Fluctuating variable
$\sim$	—	Favre averaging
$\overline{(\cdot)}$	—	Time averaging

Abbreviations

Abbreviations	Description
AVDR	Axial velocity density ratio
RANS	Reynolds-averaged Navier-Stokes
URANS	Unsteady Reynolds-averaged Navier-Stokes
LES	Large Eddy Simulation
TKE	Turbulent Kinetic Energy
LPT	Low Pressure Turbine
HPC	High Pressure Compressor
HDV	Hoch Druck Verdichter
NDT	Nieder Druck Turbine
HPC	High Performance Computing
HRLES	Hybrid RANS-LES
SS	Suction side
PS	Pressure side
LE	Leading edge
TE	Trailing edge
PSD	Power Spectral Density

Abbreviations	Description
FST	Free-Stream Turbulence
FST	Frei Strom Turbulenz
CFD	Computational Fluid Dynamics
SGS	Subgrid scale stress
Sr	Strouhal number
KH	Kelvin Helmholtz
ref	Reference
CFL	Courant-Friedrichs-Lewy
BR1	Bassi-Rebay 1
BR2	Bassi-Rebay 2
DG	Discontinuous Galerkin
DGSEM	Discontinuous Galerkin spectral element method
DNS	Direct numerical simulation
DOF	Degree of freedom
FD	Finite difference
FV	Finite volume
FE	Finite element
ILES	Implicit Large Eddy Simulation
LG	Legendre-Gauss
LGL	Legendre-Gauss-Lobatto
OP	Operating point
STG	Synthetic turbulence generator
SPOD	Spectral proper orthogonal decomposition
TRACE	Turbomachinery Research Aerodynamics Computational Environment
WALE	Wall-adapting local eddy viscosity
VB	Viscous blending
1C	One component
2C	Two component
3C	Three component
TS	Tollmien Schlichting
RS	Reynolds Stress
DLR	Deutsches Luft- und Raumfahrtzentrum
C_{ic}	Anisotropy limiting states
IP	Inflection point
PR	Prandtl number
MUSCL	Monotone upstream centered schemes

References

- Abu-Ghannam, B. J. and R. Shaw** (1980). “Natural Transition of Boundary Layers—The Effects of Turbulence, Pressure Gradient, and Flow History”. In: *Journal of Mechanical Engineering Science* 22.5, pp. 213–228. URL: https://doi.org/10.1243/JMES_JOUR_1980_022_043_02.
- Adeyinka, O. B. and G. F. Naterer** (2004). “Modeling of Entropy Production in Turbulent Flows”. In: *Journal of Fluids Engineering* 126.6, pp. 893–899. ISSN: 0098-2202. DOI: 10.1115/1.1845551.
- Akolekar, H. D., R. D. Sandberg, N. Hutchins, V. Michelassi, and G. Laskowski** (2019). “Machine-Learnt Turbulence Closures for Low-Pressure Turbines With Unsteady Inflow Conditions”. In: *Journal of Turbomachinery* 141.10. ISSN: 0889-504X. DOI: 10.1115/1.4043907.
- Akolekar, H. D., J. Weatheritt, N. Hutchins, R. D. Sandberg, G. Laskowski, and V. Michelassi** (2018). “Development and Use of Machine-Learnt Algebraic Reynolds Stress Models for Enhanced Prediction of Wake Mixing in LPTs”. In: *Proceedings of the ASME Turbo Expo: Turbomachinery Technical Conference and Exposition - 2018*. New York, N.Y.: The American Society of Mechanical Engineers. ISBN: 978-0-7918-5101-2. DOI: 10.1115/GT2018-75447.
- Alam, M. and N. D. Sandham** (2000). “Direct numerical simulation of short laminar separation bubbles with turbulent reattachment”. In: *Journal of Fluid Mechanics* 410, pp. 1–28. DOI: 10.1017/S0022112099008976.
- Altmann, C., A. D. Beck, F. Hindenlang, M. Staudenmaier, G. J. Gassner, and C. D. Munz** (2013). “An Efficient High Performance Parallelization of a Discontinuous Galerkin Spectral Element Method”. In: *Facing the Multicore-Challenge III*. Ed. by R. Keller. Vol. 7686. Lecture Notes in Computer Science Ser. Berlin, Heidelberg: Springer Berlin / Heidelberg, pp. 37–47. ISBN: 978-3-642-35892-0. DOI: 10.1007/978-3-642-35893-7_4.
- Aufderheide, T., C. Bode, J. Friedrichs, and D. Kozulovic** (2014). “The generation of higher levels of turbulence in a low-speed cascade wind tunnel by pressurized tubes”. In: *Proceedings of the 11th World Congress on Computational Mechanics*.
- Baines, W. D. and E. G. Peterson** (1951). “An Investigation of Flow Through Screens”. In: *Journal of Fluids Engineering* 73.5, pp. 467–477. ISSN: 0098-2202. DOI: 10.1115/1.4016280.

- Balzer, W.** (2011). *Numerical Investigation of the Role of Free-Stream Turbulence on Boundary-Layer Separation and Separation Control*. URL: <http://hdl.handle.net/10150/204289>.
- Balzer, W. and H. F. Fasel** (2016). “Numerical investigation of the role of free-stream turbulence in boundary-layer separation”. In: *Journal of Fluid Mechanics* 801, pp. 289–21. DOI: 10.1017/jfm.2016.424.
- Bassi, F. and S. Rebay** (1997a). “A High-Order Accurate Discontinuous Finite Element Method for the Numerical Solution of the Compressible Navier–Stokes Equations”. In: *Journal of Computational Physics* 131.2, pp. 267–279. ISSN: 0021-9991. DOI: 10.1006/jcph.1996.5572. URL: <https://www.sciencedirect.com/science/article/pii/S0021999196955722>.
- Bassi, F. and S. Rebay** (1997b). “High-Order Accurate Discontinuous Finite Element Solution of the 2D Euler Equations”. In: *Journal of Computational Physics* 138.2, pp. 251–285. ISSN: 0021-9991. DOI: 10.1006/jcph.1997.5454.
- Batchelor, G. K. and G. I. Taylor** (1946). “The theory of axisymmetric turbulence”. In: *Proceedings of the Royal Society of London. Series A. Mathematical and Physical Sciences* 186.1007, pp. 480–502. DOI: 10.1098/rspa.1946.0060.
- Beck, A. D., T. Bolemann, D. Flad, H. Frank, G. J. Gassner, F. Hindenlang, and C. D. Munz** (2014). “High-order discontinuous Galerkin spectral element methods for transitional and turbulent flow simulations”. In: *International Journal for Numerical Methods in Fluids* 76.8, pp. 522–548. DOI: 10.1002/flid.3943.
- Becker, K., K. Heitkamp, and E. Kügeler** (2010). “Recent Progress In A Hybrid-Grid CFD Solver For Turbomachinery Flows”. In: *Proceedings / CFD 2010, 5th European Conference on Computational Fluid Dynamics*. Ed. by J. C. F. Pereira, A. Sequeira, and J. M. C. Pereira. Lisbon. ISBN: 978-989-96778-1-4. URL: <https://elib.dlr.de/68938/>.
- Bergmann, M., S. Drapkina, G. Ashcroft, and C. Frey** (2016). “A comparison of various nodal discontinuous Galerkin methods for the 3d Euler equations”. In: *EC-COMAS Congress 2016*. Ed. by M. Papadrakakis, V. Papadopoulos, G. Stefanou, and V. Plevris. Athens: Institute of Structural Analysis and Antiseismic Research School of Civil Engineering National Technical University of Athens (NTUA), pp. 7956–7966. ISBN: 978-618-82844-0-1. DOI: 10.7712/100016.2389.6203.
- Bergmann, M.** (2023). “A Split-Form Discontinuous Galerkin Spectral Element Framework for Scale-Resolving Simulations of Turbomachinery Flows”. PhD thesis. Ruhr Universität Bochum. URL: <https://elib.dlr.de/202558/>.
- Bergmann, M., C. Morsbach, G. Ashcroft, and E. Kügeler** (2022). “Statistical Error Estimation Methods for Engineering-Relevant Quantities From Scale-Resolving Simulations”. In: *Journal of Turbomachinery* 144.3, p. 14. ISSN: 0889504X.

- Berkooz, G., P. Holmes, and J. L. Lumley** (1993). “The Proper Orthogonal Decomposition in the Analysis of Turbulent Flows”. In: *Annual Review of Fluid Mechanics* 25. Volume 25, 1993, pp. 539–575. DOI: 10.1146/annurev.fl.25.010193.002543. URL: <https://www.annualreviews.org/content/journals/10.1146/annurev.fl.25.010193.002543>.
- Blaisdell, G. A., E. T. Spyropoulos, and J. H. Qin** (1996). “The effect of the formulation of nonlinear terms on aliasing errors in spectral methods”. In: *Applied Numerical Mathematics* 21.3, pp. 207–219.
- Blazek, J.** (2005). *Computational fluid dynamics: Principles and applications*. 2. ed., 6. reprint. Amsterdam and Heidelberg: Elsevier. ISBN: 0080445063.
- Bode, C.** (2018). “Verbesserte Wiedergabe der Turbulenz in Turbinenströmungen und deren Einfluss auf das dreidimensionale Grenzschichtverhalten”. Dissertation. Technische Universität Braunschweig.
- Bode, C., T. Aufderheide, J. Friedrichs, and D. Kožulović** (2015). “Improved Turbulence and Transition Prediction for Turbomachinery Flows”. In: *Proceedings of the ASME International Mechanical Engineering Congress and Exposition - 2014*. New York, NY: ASME. ISBN: 978-0-7918-4642-1. DOI: 10.1115/IMECE2014-36866.
- Bode, C., T. Aufderheide, D. Kožulović, and J. Friedrichs** (2014). “The Effects of Turbulence Length Scale on Turbulence and Transition Prediction in Turbomachinery Flows”. In: *Proceedings of the ASME Turbo Expo: Turbine Technical Conference and Exposition - 2014*. Ed. by H. Hodson. New York, NY: ASME. ISBN: 978-0-7918-4561-5. DOI: 10.1115/GT2014-27026.
- Borcherding, T., C. Bode, M. Rosenzweig, P. J. Przytarski, J. Leggett, and R. D. Sandberg** (2023). “Entropy Loss Breakdown Comparison for LES and RANS of a Transonic Compressor Stage Midspan Section”. In: *ASME Turbo Expo 2023: Turbomachinery Technical Conference and Exposition*. DOI: 10.1115/GT2023-102949.
- Boussinesq, J.** (1877). *Théorie de l'Écoulement Tourbillant*. Vol. 23. Mém. Présentés par Divers Savants Acad. Sci.Inst.FR., pp. 46–50.
- Bradshaw, P.** (1967). “The turbulence structure of equilibrium boundary layers”. In: *Journal of Fluid Mechanics* 29.4, pp. 625–645. DOI: 10.1017/S0022112067001089.
- Bradshaw, P.** (1996). “Turbulence modeling with application to turbomachinery”. In: *Progress in Aerospace Sciences* 32.6, pp. 575–624. ISSN: 03760421. DOI: 10.1016/S0376-0421(96)00003-6.
- Brandt, L., P. Schlatter, and D. S. Henningson** (2004). “Transition in boundary layers subject to free-stream turbulence”. In: *Journal of Fluid Mechanics* 517, pp. 167–198. DOI: 10.1017/S0022112004000941.

- Camberos, J.** (2001). “On the construction of entropy balance equations for arbitrary thermophysical processes”. In: *39th Aerospace Sciences Meeting and Exhibit*. Reston, Virginia: American Institute of Aeronautics and Astronautics. DOI: 10.2514/6.2001-815.
- Celik, I. B., Marit Klein, M. Freitag, and Johannes Janicka** (2006). “Assessment measures for URANS/DES/LES: an overview with applications”. In: *Journal of Turbulence*. URL: <https://api.semanticscholar.org/CorpusID:121558074>.
- Chattot, J.-J. et al.** (2009). *Large Eddy Simulation for Compressible Flows*. Dordrecht: Springer Netherlands. ISBN: 978-90-481-2818-1. DOI: 10.1007/978-90-481-2819-8.
- Chen, L. W., R. Pichler, and R. D. Sandberg** (2015). “Compressible DNS of a Low Pressure Turbine Subjected to Inlet Disturbances”. In: *Direct and Large-Eddy Simulation IX*. Ed. by V. Armenio. Vol. 20. ERCOFTAC Ser. Cham: Springer International Publishing AG, pp. 383–388. ISBN: 978-3-319-14447-4. DOI: 10.1007/978-3-319-14448-1{\textunderscore}48.
- Choi, H. and P. Moin** (2012). “Grid-point requirements for large eddy simulation: Chapman’s estimates revisited”. In: *Physics of Fluids* 24.1, p. 011702. ISSN: 1070-6631. DOI: 10.1063/1.3676783.
- Choi, K. . and J. L. Lumley** (2001). “The return to isotropy of homogeneous turbulence”. In: *Journal of Fluid Mechanics* 436, pp. 59–84. ISSN: 0022-1120. DOI: 10.1017/S002211200100386X.
- Comte, F. D. P. and M. Lesieur** (1996). “Large-eddy simulation of transition to turbulence in a boundary layer developing spatially over a flat plate”. In: *Journal of Fluid Mechanics* 326, pp. 1–36. ISSN: 0022-1120. DOI: 10.1017/S0022112096008221.
- Cui, J., V. Nagabhushana Rao, and P. Tucker** (2016). “Numerical Investigation of Contrasting Flow Physics in Different Zones of a High-Lift Low-Pressure Turbine Blade”. In: *Journal of Turbomachinery* 138.1. ISSN: 0889-504X. DOI: 10.1115/1.4031561.
- Cui, J., V. Nagabhushana Rao, and P.G. Tucker** (2017). “Numerical investigation of secondary flows in a high-lift low pressure turbine”. In: *International Journal of Heat and Fluid Flow* 63, pp. 149–157. DOI: 10.1016/j.ijheatfluidflow.2016.05.018.
- Curtis, E. M., H. P. Hodson, M. R. Banieghbal, J. D. Denton, R. J. Howell, and N. W. Harvey** (1997). “Development of Blade Profiles for Low-Pressure Turbine Applications”. In: *Journal of Turbomachinery* 119.3, pp. 531–538. ISSN: 0889-504X. DOI: 10.1115/1.2841154.
- Denton, J. D.** (1993). “The 1993 IGTI Scholar Lecture: Loss Mechanisms in Turbomachines”. In: *Journal of Turbomachinery* 115.4, pp. 621–656. ISSN: 0889-504X. DOI: 10.1115/1.2929299.

- Denton, J. D.** (2010). “Some Limitations of Turbomachinery CFD”. In: *Proceedings of the ASME Turbo Expo 2010*. New York, NY: ASME, pp. 735–745. ISBN: 978-0-7918-4402-1. DOI: 10.1115/GT2010-22540.
- Deshpande, R. and R. Vinuesa** (2024). “Streamwise energy-transfer mechanisms in zero- and adverse-pressure-gradient turbulent boundary layers”. In: *Journal of Fluid Mechanics* 997, A16 <div></div>. DOI: 10.1017/jfm.2024.816.
- De Vincentiis, L., K. Đurović, D. Lengani, D. Simoni, J. Pralits, D. S. Henningson, and A. Hanifi** (2023). “Effects of Upstream Wakes on the Boundary Layer Over a Low-Pressure Turbine Blade”. In: *Journal of Turbomachinery* 145.5. ISSN: 0889-504X. DOI: 10.1115/1.4056108.
- Diwan, S. S. and O. N. Ramesh** (2009). “On the origin of the inflectional instability of a laminar separation bubble”. In: *Journal of Fluid Mechanics* 629, pp. 263–298. DOI: 10.1017/S002211200900634X.
- Donovan, M. H.** (2023). “Impact of Free-Stream Turbulence Intensity on the Endwall Region of Low Pressure Turbine Blades”. PhD thesis. University of Dayton. URL: http://rave.ohiolink.edu/etdc/view?acc_num=dayton1681823544653736.
- Dunham, J.** (1991). “Compressor Aerodynamics. N.A. Cumpsty. Longman Scientific & Technical, Longman Group, Longman House, Burnt Mill, Harlow CM20 2JE. 1989. 509 pp. Illustrated. £49.00”. In: *Aeronautical Journal*. URL: <https://www.semanticscholar.org/paper/Compressor-Aerodynamics.-N.A.-Cumpsty.-Longman-%26-Dunham/a1d71a8672c131ccd9583af2b0bf12b086ec26e1>.
- Duraisamy, K.** (2021). “Perspectives on machine learning-augmented Reynolds-averaged and large eddy simulation models of turbulence”. In: *Physical Review Fluids* 6.5. DOI: 10.1103/PhysRevFluids.6.050504.
- Durbin, P. and B. Reif** (2010). *Statistical Theory and Modeling for Turbulent Flows*. second. Chichester, UK: Wiley. ISBN: 978-0-470-68931-8.
- Đurović, K., L. de Vincentiis, D. Simoni, D. Lengani, J. Pralits, D. S. Henningson, and A. Hanifi** (2021). “Free-Stream Turbulence-Induced Boundary-Layer Transition in Low-Pressure Turbines”. In: *Journal of Turbomachinery* 143.8. ISSN: 0889-504X. DOI: 10.1115/1.4050450. URL: <https://asmedigitalcollection.asme.org/turbomachinery/article/143/8/081015/1103265/Free-Stream-Turbulence-Induced-Boundary-Layer>.
- Edmund Kügeler** (2005). “Numerisches Verfahren zur genauen Analyse der Kühleffektivität filmgekühlter Turbinenschaufeln”. Doctoral thesis. Ruhr-Universität Bochum, Universitätsbibliothek.
- Einstein, A.** (1916). “Die Grundlage der allgemeinen Relativitätstheorie”. In: *Annalen der Physik* 354.7, pp. 769–822. DOI: 10.1002/andp.19163540702.

- Emmons, H. W.** (1951). “The Laminar-Turbulent Transition in a Boundary Layer-Part I”. In: *Journal of the Aeronautical Sciences* 18.7, pp. 490–498. DOI: 10.2514/8.2010.
- Emory, M. and G. Iaccarino** (2014). *Visualizing turbulence anisotropy in the spatial domain with componentality contours*. Annual research briefs 2014. Stanford, CA, USA: Center for Turbulence Research, Stanford University.
- Entlesberger, R. G., M. Martinstetter, and W. Staudacher** (2005). *Untersuchungen am Turbinengitter T161 zur Bestimmung der Profildruckverteilung und der Gittercharakteristik*. Tech. rep. LRT-WE12-05/02. Universität der Bundeswehr München.
- Fang, Y., Y. Zhao, H. D. Akolekar, A. S. H. Ooi, R. D. Sandberg, R. Pacciani, and M. Marconcini** (2024). “A Data-Driven Approach for Generalizing the Laminar Kinetic Energy Model for Separation and Bypass Transition in Low- and High-Pressure Turbines”. In: *Journal of Turbomachinery* 146.9. ISSN: 0889-504X. DOI: 10.1115/1.4065124.
- Fard Afshar, N., D. Kozulovic, S. Henninger, J. Deutsch, and P. Bechlars** (2023a). “Turbulence anisotropy analysis at the middle section of a highly loaded 3D linear turbine cascade using Large Eddy Simulation”. In: *Journal of the Global Power and Propulsion Society* 7, pp. 71–84. DOI: 10.33737/jgpps/159784.
- Fard Afshar, N., F. M. Möller, J. Deutsch, S. Henninger, C. Morsbach, D. Kozulovic, P. Bechlars, and P. Jeschke** (2024). “Prediction and Analysis of Turbulence Anisotropy in a Low-Pressure Turbine Cascade at Two Reynolds Numbers Based on Transitional DDES”. In: *Proceedings of ASME Turbo Expo 2024: Turbomachinery Technical Conference and Exposition (GT2024)*. Little Falls, NJ, USA: The American Society of Mechanical Engineers. DOI: 10.1115/GT2024-121415. URL: <https://elib.dlr.de/206161/>.
- Fard Afshar, N., F. M. Möller, S. Henninger, D. Kozulovic, C. Morsbach, P. Bechlars, and P. Jeschke** (2023b). “Assessment of the DDES- γ model for the simulation of a highly loaded turbine cascade”. In: *Journal of Turbomachinery*, pp. 1–12. ISSN: 0889-504X. DOI: 10.1115/1.4064079.
- Ferziger, J. H.** (2008). *Numerische Strömungsmechanik*. Berlin, Heidelberg: Springer Berlin / Heidelberg. ISBN: 9783540682288. URL: <https://ebookcentral.proquest.com/lib/kxp/detail.action?docID=6438383>.
- Fischberg, C. J., C. M. Rhie, R. M. Zacharias, P. C. Bradley, and T. M. DesSureault** (1996). “Using hundreds of workstations for production running of parallel CFD applications”. In: URL: <https://api.semanticscholar.org/CorpusID:58738488>.
- Gao, F., G. Zambonini, J. Boudet, X. Ottavy, L. Lu, and L. Shao** (2015). “Unsteady behavior of corner separation in a compressor cascade: Large eddy simulation and experimental study”. In: *Proceedings of the Institution of Mechanical Engineers Part A Journal of Power and Energy*. DOI: 10.1177/0957650915594314.

- Gaster, M.** (1969). *The structure and behaviour of laminar separation bubbles*. Vol. 3595. Reports and memoranda / Aeronautical Research Council. London, UK: HMSO. ISBN: 011470192X.
- Georgiadis, N. J., D. P. Rizzetta, and C. Fureby** (2010). “Large-Eddy Simulation: Current Capabilities, Recommended Practices, and Future Research”. In: *AIAA Journal* 48.8, pp. 1772–1784. DOI: 10.2514/1.J050232.
- Geurts, B. J. and J. Fröhlich** (2002). “A framework for predicting accuracy limitations in large-eddy simulation”. In: *Physics of Fluids* 14.6, pp. L41–L44. ISSN: 1070-6631. DOI: 10.1063/1.1480830.
- Goodhand, M. N., R. J. Miller, and H. W. Lung**, eds. (2012). *The Sensitivity of 2D Compressor Incidence Range to In-Service Geometric Variation*. Vol. Volume 8: Turbomachinery, Parts A, B, and C. Turbo Expo: Power for Land, Sea, and Air. DOI: 10.1115/GT2012-68633.
- Gourdain, N., F. Sicot, F. Duchaine, and L. Gicquel** (2014). “Large eddy simulation of flows in industrial compressors: a path from 2015 to 2035”. In: *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences* 372.2022, p. 20130323. DOI: 10.1098/rsta.2013.0323.
- Graveline, S. and S. Sjolander** (2005). “Spectral Analysis of Measurements of Separated Flow Transition on a Turbine Airfoil”. In: *35th AIAA Fluid Dynamics Conference and Exhibit*. Reston, Va.: American Institute of Aeronautics and Astronautics. ISBN: 978-1-62410-059-8. DOI: 10.2514/6.2005-4769.
- Gregor J. Gassner, Andrew R. Winters, and David A. Kopriva** (2016). “Split form nodal discontinuous Galerkin schemes with summation-by-parts property for the compressible Euler equations”. In: *Journal of Computational Physics* 327, pp. 39–66. ISSN: 0021-9991. DOI: 10.1016/j.jcp.2016.09.013. URL: <https://www.sciencedirect.com/science/article/pii/S0021999116304259>.
- Halstead, D. E., D. C. Wisler, H. W. Shin, T. H. Okiishi, G. J. Walker, and H. P. Hodson** (1997). “Boundary layer development in axial compressors and turbines. Part 2 of 4: Compressors”. In: *Journal of Turbomachinery* 119.3. ISSN: 0889-504X. DOI: 10.1115/1.2841142.
- Hammond, J., N. Pepper, F. Montomoli, and V. Michelassi** (2022). “Machine Learning Methods in CFD for Turbomachinery: A Review”. In: *International Journal of Turbomachinery, Propulsion and Power* 7.2. ISSN: 2504-186X. DOI: 10.3390/ijtp7020016. URL: <https://www.mdpi.com/2504-186X/7/2/16>.
- Hilgenfeld, L. and M. Pfitzner** (2004). “Unsteady Boundary Layer Development Due to Wake Passing Effects on a Highly Loaded Linear Compressor Cascade”. In: *Journal of Turbomachinery* 126.4, pp. 493–500. ISSN: 0889-504X. DOI: 10.1115/1.1791290.

- Himmel, C. G.** (2010). “Ultra-high lift blades for low pressure turbines”. Thesis. University of Cambridge.
- Hindenlang, F., G. J. Gassner, C. Altmann, A. Beck, M. Staudenmaier, and C. D. Munz** (2012). “Explicit discontinuous Galerkin methods for unsteady problems”. In: *Computers and Fluids* 61.Complete, pp. 86–93. DOI: 10.1016/j.compfluid.2012.03.006.
- Hirsch, C.** (2007). *Numerical Computation of Internal and External Flows: The Fundamentals of Computational Fluid Dynamics*. 2nd ed. San Diego: Elsevier Science & Technology. ISBN: 9780080550022. URL: <https://ebookcentral.proquest.com/lib/kxp/detail.action?docID=7266341>.
- Hodson, H. P. and R. J. Howell** (2005a). “Bladerow interactions, transition, and high-lift aerofoils in low-pressure turbines”. In: *Annual Review of Fluid Mechanics* 37.1, pp. 71–98. DOI: 10.1146/annurev.fluid.37.061903.175511.
- Hodson, H. P. and R. J. Howell** (2005b). “The role of transition in high-lift low-pressure turbines for aeroengines”. In: *Progress in Aerospace Sciences* 41.6, pp. 419–454. ISSN: 03760421. DOI: 10.1016/j.paerosci.2005.08.001.
- Horton, H. P.** (1968). “Laminar separation bubbles in two and three dimensional incompressible flow”. In: URL: <https://api.semanticscholar.org/CorpusID:107046848>.
- Hösgen, C.** (2019). “Analyse des turbulenten Nachlaufs eines Verdichtergitters”. Dissertation. Rheinisch-Westfälische Technische Hochschule Aachen and Verlag Dr. Hut.
- Hosseini, S. M., D. Simoni, and A. Hanifi** (2015). *Direct numerical simulation of flow around a turbine blade: A transition study*. URL: <https://www.diva-portal.org/smash/record.jsf?pid=diva2:873661>.
- Hourmouziadis, J.** (1989). “Aerodynamic Design of Low-Pressure Turbines”. In: *Blading Design for Axial Turbomachines*. AGARD Lecture Series 167.
- Jacobs, R. G. and P. A. Durbin** (1998). “Shear sheltering and the continuous spectrum of the Orr–Sommerfeld equation”. In: *Physics of Fluids* 10.8, pp. 2006–2011. ISSN: 1070-6631. DOI: 10.1063/1.869716.
- Jardine, L.** (2019). “The Effect of Heat Transfer on Turbine Performance: The Effect of Heat Transfer on Turbine Performance”. PhD thesis. DOI: 10.17863/CAM.56517. URL: <https://www.repository.cam.ac.uk/items/7f803ffd-26e4-490b-ae21-c51243a0178b>.
- Jeong, H. and S. J. Song** (2024). “Impact of Low Freestream Turbulence and Surface Roughness on the Flat-Plate Boundary Layer Transition Under a High-Lift Airfoil Pressure Gradient”. In: *Journal of Turbomachinery* 146.11. ISSN: 0889-504X. DOI: 10.1115/1.4065494.

- Jones, W. and B. Launder** (1972). “The prediction of laminarization with a two-equation model of turbulence”. In: *International Journal of Heat and Mass Transfer* 15.2, pp. 301–314. ISSN: 0017-9310. DOI: 10.1016/0017-9310(72)90076-2.
- Kalitzin, G., X. Wu, and P. A. Durbin** (2003). “DNS of fully turbulent flow in a LPT passage”. In: *International Journal of Heat and Fluid Flow* 24.4, pp. 636–644. DOI: 10.1016/S0142-727X(03)00057-2. URL: <https://www.sciencedirect.com/science/article/pii/S0142727X03000572>.
- Kato, M. and B. E. Launder** (1993). “The Modelling of Turbulent Flow Around Stationary and Vibrating Square Cylinders”. In: *9th Symposium on Turbulent Shear Flows*. Ed. by F. Durst, N. Kasagi, B. E. Launder, F. W. Schmidt, K. Suzuki, and J. H. Whitelaw. Berlin, Heidelberg: Springer Berlin Heidelberg, pp. 10.4.1–10.4.6. ISBN: 978-3-642-78825-3.
- Klebanoff, P. S., K. D. Tidstrom, and L. M. Sargent** (1962). “The three-dimensional nature of boundary-layer instability”. In: *Journal of Fluid Mechanics* 12.1, pp. 1–34. DOI: 10.1017/S0022112062000014.
- Kopriva, D. A.** (2009). *Implementing Spectral Methods for Partial Differential Equations*. Dordrecht: Springer Netherlands. ISBN: 978-90-481-2260-8. DOI: 10.1007/978-90-481-2261-5.
- Kopriva, D. A., S. L. Woodruff, and M. Y. Hussaini** (2002). “Computation of electromagnetic scattering with a nonconforming discontinuous spectral element method”. In: *International Journal for Numerical Methods in Engineering* 53, pp. 105–122. DOI: 10.1002/nme.394.
- Koschichow, D., J. Fröhlich, I. Kirik, and R. Niehuis** (2014). “DNS of the Flow Near the Endwall in a Linear Low Pressure Turbine Cascade With Periodically Passing Wakes”. In: *Proceedings of the ASME Turbo Expo: Turbine Technical Conference and Exposition - 2014*. Ed. by H. Hodson. New York, NY: ASME. ISBN: 978-0-7918-4563-9. DOI: 10.1115/GT2014-25071.
- Kožulović, D.** (2008). “Modellierung des Grenzschichtumschlags bei Turbomaschinenströmungen unter Berücksichtigung mehrerer Umschlagsarten”. Doctoral thesis. Ruhr-Universität Bochum, Universitätsbibliothek.
- Krause, M., M. Behr, and J. Ballmann** (2008). “Modeling of Transition Effects in Hypersonic Intake Flows Using a Correlation-Based Intermittency Model”. In: *15th AIAA International Space Planes and Hypersonic Systems and Technologies Conference*. Reston, Virginia: American Institute of Aeronautics and Astronautics. ISBN: 978-1-60086-985-3. DOI: 10.2514/6.2008-2598.
- Lang, H. M.** (2024). “Einfluss der turbulenten Abklingrate auf die Umströmung eines Verdichtergitters”. PhD thesis. RWTH Aachen University. DOI: 10.18154/RWTH-2024-05660.

- Langtry, R. B. and F. R. Menter** (2009). “Correlation-Based Transition Modeling for Unstructured Parallelized Computational Fluid Dynamics Codes”. In: *AIAA Journal* 47.12, pp. 2894–2906. ISSN: 0001-1452. DOI: 10.2514/1.42362.
- Langtry, R. B.** (2006). “A correlation-based transition model using local variables for unstructured parallelized CFD codes”. PhD thesis. DOI: 10.18419/opus-1705.
- Laskowski, G. M., J. Kopriva, V. Michelassi, S. Shankaran, U. Paliath, R. Bhaskaran, Q. Wang, C. Talnikar, Z. J. Wang, and F. Jia** (2016). “Future Directions of High Fidelity CFD for Aerothermal Turbomachinery Analysis and Design”. In: *46th AIAA Fluid Dynamics Conference*. DOI: 10.2514/6.2016-3322.
- Launder, B. E. and N. D. Sandham** (2002). *Closure strategies for turbulent and transitional flows*. 1st ed. Cambridge, UK: Cambridge University Press. ISBN: 1-107-12984-2.
- Launder, B. and D. Spalding** (1974). “The numerical computation of turbulent flows”. In: *Computer Methods in Applied Mechanics and Engineering* 3.2, pp. 269–289. ISSN: 0045-7825. DOI: 10.1016/0045-7825(74)90029-2.
- Leggett, J., E. Richardson, S. Priebe, A. Shabbir, V. Michelassi, and R. Sandberg** (2020). “Loss Analysis of Unsteady Turbomachinery Flows Based on the Mechanical Work Potential”. In: *Journal of Turbomachinery* 142.11. ISSN: 0889-504X. DOI: 10.1115/1.4048162.
- Leggett, J., Y. Zhao, E. S. Richardson, and R. D. Sandberg** (2021). “Turbomachinery Loss Analysis: The Relationship Between Mechanical Work Potential and Entropy Analyses”. In: *Proceedings of ASME Turbo Expo 2021: Turbomachinery Technical Conference and Exposition*. New York, N.Y.: The American Society of Mechanical Engineers. ISBN: 978-0-7918-8492-8. DOI: 10.1115/GT2021-59436.
- Lei, V.-M., Z. S. Spakovszky, and E. M. Greitzer** (2008). “A Criterion for Axial Compressor Hub-Corner Stall”. In: *Journal of Turbomachinery* 130.3. ISSN: 0889-504X. DOI: 10.1115/1.2775492.
- Lengani, D., D. Simoni, J. O. Pralits, K. Đurović, L. de Vincentiis, D. S. Henningson, and A. Hanifi** (2022). “On the receptivity of low-pressure turbine blades to external disturbances”. In: *Journal of Fluid Mechanics* 937. DOI: 10.1017/jfm.2022.127.
- Lengani, D., D. Simoni, M. Ubaldi, P. Zunino, and F. Bertini** (2017). “A POD-Based Procedure for the Split of Unsteady Losses of an LPT Cascade”. In: *International Journal of Turbomachinery, Propulsion and Power* 2.4. ISSN: 2504-186X. DOI: 10.3390/ijtp2040017. URL: <https://www.mdpi.com/2504-186X/2/4/17>.
- Leschziner, M.** (2015). *Statistical Turbulence Modelling for Fluid Dynamics - Demystified*. IMPERIAL COLLEGE PRESS. DOI: 10.1142/p997.

- LeVeque, R. J.** (2012). *Finite Volume Methods for Hyperbolic Problems*. Cambridge, UK: Cambridge University Press. ISBN: 9780521810876. DOI: 10.1017/CB09780511791253.
- Leyh, S. and C. Morsbach** (2020). “The Coupling of a Synthetic Turbulence Generator with Turbomachinery Boundary Conditions”. In: *Direct and Large Eddy Simulation XII*. Ed. by M. García-Villalba. Vol. 27. ERCOFTAC Ser. Cham: Springer International Publishing AG, pp. 349–355. ISBN: 978-3-030-42821-1. DOI: 10.1007/978-3-030-42822-8\textunderscore46.
- Ludewig, T., M. Mack, R. Niehuis, and M. Franke** (Jan. 2011). “Optimization of the blowing ratio for a lowpressure turbine cascade with active flow control”. In: *9th European Conference on Turbomachinery: Fluid Dynamics and Thermodynamics, ETC 2011 - Conference Proceedings 2*.
- Lumley, J. L.** (1979). “Computational Modeling of Turbulent Flows”. In: *Advances in Applied Mechanics Volume 18*. Vol. 18. Advances in Applied Mechanics. Elsevier, pp. 123–176. ISBN: 9780120020188. DOI: 10.1016/S0065-2156(08)70266-7.
- Malan, P., K. Suluksna, and E. Juntasaro** (2012). “Calibrating the Gamma-Retheta Transition Model for Commercial CFD”. In: *47th AIAA Aerospace Sciences Meeting including The New Horizons Forum and Aerospace Exposition*. DOI: 10.2514/6.2009-1142.
- Marsden, J. E., L. Sirovich, P. J. Schmid, and D. S. Henningson** (2001). *Stability and Transition in Shear Flows*. Vol. 142. New York, NY: Springer New York. ISBN: 978-1-4612-6564-1. DOI: 10.1007/978-1-4613-0185-1.
- Martinstetter, M.** (2010). “Experimentelle Untersuchungen zur Aerodynamik hoch belasteter Niederdruckturbinen-Beschaufelungen”. Dissertation. Neubiberg: Universität der Bundeswehr München, Fakultät für Luft- und Raumfahrttechnik.
- Matsuura, K. and C. Kato** (2006). “Large-Eddy Simulation of Compressible Transitional Cascade Flows with and without Incoming Free-Stream Turbulence”. In: *JSME International Journal Series B Fluids and Thermal Engineering* 49.3, pp. 660–669. DOI: 10.1299/jsmeb.49.660.
- Mayle, R. E.** (1991). “The 1991 IGTI Scholar Lecture: The Role of Laminar-Turbulent Transition in Gas Turbine Engines”. In: *Journal of Turbomachinery* 113.4, pp. 509–536. ISSN: 0889-504X. DOI: 10.1115/1.2929110.
- Mayle, R. E.** (2018). *Elements of Transitional Boundary-Layer Flow*. Second Enhanced Edition, revidierte Ausgabe. Berlin: Logos Berlin. ISBN: 9783832545987.
- McAuliffe, B. R. and M. I. Yaras** (2008). “Numerical Study of Turbulent-Spot Development in a Separated Shear Layer”. In: *Journal of Turbomachinery* 130.4, p. 041018. ISSN: 0889-504X. DOI: 10.1115/1.2812948.

- McAuliffe, B. R. and M. I. Yaras** (2010). “Transition Mechanisms in Separation Bubbles Under Low- and Elevated-Freestream Turbulence”. In: *Journal of Turbomachinery* 132.1. ISSN: 0889-504X. DOI: 10.1115/1.2812949.
- Medic, G. and P. A. Durbin** (2002). “Toward Improved Prediction of Heat Transfer on Turbine Blades”. In: *Journal of Turbomachinery* 124.2, pp. 187–192. ISSN: 0889-504X. DOI: 10.1115/1.1458020.
- Mengaldo, G. and R. Maulik** (2021). “PySPOD: A Python package for Spectral Proper Orthogonal Decomposition (SPOD)”. In: *Journal of Open Source Software* 6.60, p. 2862. DOI: 10.21105/joss.02862.
- Menter, F. R.** (1994). “Two-equation eddy-viscosity turbulence models for engineering applications”. In: *AIAA Journal* 32.8, pp. 1598–1605. ISSN: 0001-1452. DOI: 10.2514/3.12149.
- Menter, F. R., A. Matyushenko, and R. Lechner** (2020). “Development of a Generalized $K-\omega$ Two-Equation Turbulence Model”. In: *New results in numerical and experimental fluid mechanics XII*. Ed. by A. Dillmann, G. Heller, E. Krämer, C. Wagner, C. Tropea, and S. Jakirlić. Vol. 142. Notes on Numerical Fluid Mechanics and Multidisciplinary Design. Cham, Switzerland: springer, pp. 101–109. ISBN: 978-3-030-25252-6. DOI: 10.1007/978-3-030-25253-3\textunderscore10.
- Menter, F. R., P. E. Smirnov, T. Liu, and R. Avancha** (July 2015). “A One-Equation Local Correlation-Based Transition Model”. In: *Flow, Turbulence and Combustion* 95.4, pp. 583–619. DOI: 10.1007/s10494-015-9622-4.
- Menter, F. R., R. Langtry, S. Likki, Y. B. Suzen, P. G. Huang, and S. Völker** (2004). “A Correlation-Based Transition Model Using Local Variables: Part I — Model Formulation”. In: URL: <https://api.semanticscholar.org/CorpusID:118446774>.
- Michelassi, V., L. W. Chen, R. Pichler, and R. D. Sandberg** (2015). “Compressible Direct Numerical Simulation of Low-Pressure Turbines—Part II: Effect of Inflow Disturbances”. In: *Journal of Turbomachinery* 137.7. ISSN: 0889-504X. DOI: 10.1115/1.4029126.
- Michelassi, V., J. Wissink, and W. Rodi** (2002). “Analysis of DNS and LES of Flow in a Low Pressure Turbine Cascade with Incoming Wakes and Comparison with Experiments”. In: *Flow, Turbulence and Combustion* 69.3, pp. 295–329. ISSN: 1573-1987. DOI: 10.1023/A:1027334303200. URL: <https://doi.org/10.1023/A:1027334303200>.
- Miller, R. J.** (2013). “Mechanical Work Potential”. In: *Proceedings of the ASME Turbo Expo: Turbine Technical Conference and Exposition - 2013*. Ed. by S. J. Song. New York, N.Y: ASME. ISBN: 978-0-7918-5522-5. DOI: 10.1115/GT2013-95488.

- Monier, J.-F., F. Gao, J. Boudet, and L. Shao** (2020). “Turbulence modelling analysis in a corner separation flow”. In: *Computers & Fluids* 213, p. 104745. ISSN: 00457930. DOI: 10.1016/j.compfluid.2020.104745.
- Morkovin, M. V.** (1993). “Bypass-Transition Research: Issues and Philosophy”. In: *Instabilities and Turbulence in Engineering Flows*. Ed. by D. E. Ashpis, T. B. Gatski, and R. Hirsh. Fluid Mechanics and Its Applications. Dordrecht: springer, pp. 3–30. ISBN: 978-94-011-1743-2. DOI: 10.1007/978-94-011-1743-2_1.
- Morsbach, C., M. Bergmann, A. Tosun, B. F. Klose, E. Kügeler, and M. Franke** (2023). “Large Eddy Simulation of a Low-Pressure Turbine Cascade with Turbulent End Wall Boundary Layers”. In: *Flow, Turbulence and Combustion*, p. 28. ISSN: 1573-1987. DOI: 10.1007/s10494-023-00502-6. URL: <https://doi.org/10.1007/s10494-023-00502-6>.
- Morsbach, C.** (2017). “Reynolds Stress Modelling for Turbomachinery Flow Applications”. Dissertation. Darmstadt, Germany: German Aerospace Center, Technische Universität Darmstadt.
- Morsbach, C. and M. Bergmann** (2020). “Critical Analysis of the Numerical Setup for the Large-Eddy Simulation of the Low-Pressure Turbine Profile T106C”. In: *Direct and Large Eddy Simulation XII*. Ed. by M. García-Villalba. Vol. 27. ERCOFTAC Ser. Cham: Springer International Publishing AG, pp. 343–348. ISBN: 978-3-030-42821-1. DOI: 10.1007/978-3-030-42822-8{\\textunderscore}45.
- Müller-Schindewolffs, C.** (2022). “Entwicklung eines Modells zur Vorhersage stationärer Transitionsmechanismen in Niederdruckturbinen unter dem Einsatz von skalenauflösenden Simulationen”. In: *Berichte aus dem Institut für Turbomaschinen und Fluid-Dynamik*;28. DOI: 10.15488/11982. URL: <https://www.repo.uni-hannover.de/handle/123456789/12079>.
- Nicoud, F. and F. Ducros** (1999). “Subgrid-scale stress modelling based on the square of the velocity gradient tensor”. In: *Flow, Turbulence and Combustion* 62.3, pp. 183–200. ISSN: 1386-6184.
- Pichler, R., R. Sandberg, V. Michelassi, and R. Bhaskaran** (2015). “Investigation of the Accuracy of RANS Models to Predict the Flow Through a Low-Pressure Turbine”. In: *Proceedings of the ASME Turbo Expo: Turbine Technical Conference and Exposition - 2015*. New York, NY: ASME. ISBN: 978-0-7918-5664-2. DOI: 10.1115/GT2015-43446.
- Pichler, R., R. D. Sandberg, and V. Michelassi** (2016). “Assessment of Grid Resolution Requirements for Accurate Simulation of Disparate Scales of Turbulent Flow in Low-Pressure Turbines”. In: *Proceedings of the ASME Turbo Expo: Turbine Technical Conference and Exposition - 2016*. New York, N.Y.: The American Society of Mechanical Engineers. ISBN: 978-0-7918-4971-2. DOI: 10.1115/GT2016-56858.

- Pichler, R., Y. Zhao, R. D. Sandberg, V. Michelassi, R. Pacciani, M. Marconcini, and A. Arnone (2018). “LES and RANS Analysis of the End-Wall Flow in a Linear LPT Cascade: Part I — Flow and Secondary Vorticity Fields Under Varying Inlet Condition”. In: *Proceedings of the ASME Turbo Expo: Turbomachinery Technical Conference and Exposition - 2018*. New York, N.Y.: The American Society of Mechanical Engineers. ISBN: 978-0-7918-5100-5. DOI: 10.1115/GT2018-76233.
- Pino Martín, M., U. Piomelli, and G. V. Candler (2000). “Subgrid-Scale Models for Compressible Large-Eddy Simulations”. In: *Theoretical and Computational Fluid Dynamics* 13.5, pp. 361–376. DOI: 10.1007/PL00020896.
- Pope, S. B. (2000). *Turbulent Flows*. Cambridge, UK: Cambridge University Press.
- Przytarski, P. J., D. Lengani, D. Simoni, and A. P. S. Wheeler (2024). “The Role of Turbulence Transport in Mechanical Energy Budgets”. In: *Journal of Turbomachinery* 146.10. ISSN: 0889-504X. DOI: 10.1115/1.4065287.
- Przytarski, P. J. and A. P. S. Wheeler (2020). “The Effect of Gapping on Compressor Performance”. In: *Journal of Turbomachinery* 142.12. ISSN: 0889-504X. DOI: 10.1115/1.4047933.
- Przytarski, P. J. and A. P. S. Wheeler (2021). “Accurate Prediction of Loss Using High Fidelity Methods”. In: *Journal of Turbomachinery* 143.3. ISSN: 0889-504X. DOI: 10.1115/1.4050115.
- Raverdy, B., I. Mary, P. Sagaut, and N. Liamis (2003). “High-Resolution Large-Eddy Simulation of Flow Around Low-Pressure Turbine Blade”. In: *AIAA Journal* 41.3, pp. 390–397. DOI: 10.2514/2.1989.
- Reynolds, O. (1895). “IV. On the dynamical theory of incompressible viscous fluids and the determination of the criterion”. In: *Philosophical Transactions of the Royal Society of London. (A.)* 186, pp. 123–164. ISSN: 0264-3820. DOI: 10.1098/rsta.1895.0004.
- Rezaeiravesh, S. and M. Liefvendahl (2018). “Effect of grid resolution on large eddy simulation of wall-bounded turbulence”. In: *Physics of Fluids* 30.5, p. 055106. ISSN: 1070-6631. DOI: 10.1063/1.5025131.
- Richardson, L. F. (1922). *Weather prediction by numerical process*. Cambridge Mathematical Library 2. Cambridge, United Kingdom: Cambridge University Press. ISBN: 9780521680448. URL: <https://archive.org/details/weatherprediction00richrich>.
- Röber, T., D. Kozulovic, E. Kügeler, and D. Nürnberger (2005). “Appropriate Turbulence Modelling for Turbomachinery Flows”. In: *Strömungen mit Ablösung, 14. DGLR-Fach-Symposium Bremen, 14. - 16. November 2004*. Notes on Numerical Fluid Mechanics and Multidisciplinary Design. LIDO-Berichtsjahr=2005, Springer Verlag. URL: <https://elib.dlr.de/1301/>.

- Robison, Z. and A. Gross** (2021). “Large-Eddy Simulation of Low-Pressure Turbine Cascade with Unsteady Wakes”. In: *Aerospace* 8.7, p. 184. DOI: 10.3390/aerospace 8070184.
- Roe, P.** (1981). “Approximate Riemann solvers parameter vectors and difference schemes”. In: *Journal of Computational Physics* 43.2, pp. 357–372.
- Rosenzweig, M., M. Kozul, R. Sandberg, M. Marconcini, and R. Pacciani** (2023). “High-Fidelity Simulations and RANS Analysis of a Low-Pressure Turbine Cascade in a Spanwise-Diverging Gas Path - End-Wall Flow Analysis and Loss Generation Mechanisms”. In: *Volume 13B: Turbomachinery — Axial Flow Turbine Aerodynamics*. American Society of Mechanical Engineers. ISBN: 978-0-7918-8709-7. DOI: 10.1115/GT2023-101039.
- Sagaut, P.** (2001). “LES and DNS simulation of turbomachinery flows”. In: VKI Lecture Series - Recent Developments in Numerical Methods for Turbomachinery Flows. ed. by T. Arts. Von Karman Institute for Fluid Dynamics.
- Sagaut, P.** (2006). *Large Eddy simulation for incompressible flows: An introduction*. Soft-cover reprint of the hardcover 3. ed. Physics and astronomy online library. Berlin and Heidelberg: springer. ISBN: 3642065805.
- Sandberg, R. D. and V. Michelassi** (2019). “The Current State of High-Fidelity Simulations for Main Gas Path Turbomachinery Components and Their Industrial Impact”. In: *Flow, Turbulence and Combustion* 102.4, pp. 797–848. ISSN: 13866184. DOI: 10.1007/s10494-019-00013-3.
- Sandberg, R. D., V. Michelassi, R. Pichler, L. Chen, and R. Johnstone** (2015). “Compressible Direct Numerical Simulation of Low-Pressure Turbines-Part I: Methodology”. In: *Journal of Turbomachinery* 137.5. ISSN: 0889-504X. DOI: 10.1115/1.4028731.
- Saric, W. S., H. L. Reed, and E. J. Kerschen** (2002). “Boundary-layer receptivity to freestream disturbances”. In: *Annual Review of Fluid Mechanics* 34, pp. 291–319. ISSN: 0066-4189. DOI: 10.1146/annurev.fluid.34.082701.161921.
- Sarkar, S.** (2007). “Effects of passing wakes on a separating boundary layer along a low-pressure turbine blade through large-eddy simulation”. In: *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* 221.4, pp. 551–564. DOI: 10.1243/09576509JPE400.
- Sarkar, S.** (2008). “Identification of Flow Structures on a LP Turbine Blade Due to Periodic Passing Wakes”. In: *Journal of Fluids Engineering* 130.6. ISSN: 0098-2202. DOI: 10.1115/1.2911682.
- Sarkar, S.** (2009). “Influence of Wake Structure on Unsteady Flow in a Low Pressure Turbine Blade Passage”. In: *Journal of Turbomachinery* 131.4. ISSN: 0889-504X. DOI: 10.1115/1.3072490.

- Schlatter, P.** (2012). *Boundary Layer DNS/LES Data*. URL: <https://www.mech.kth.se/~pschlatt/DATA/> (visited on 05/30/2014).
- Schlichting, H. and K. Gersten** (2017). *Boundary-Layer Theory*. Berlin, Heidelberg: Springer Berlin Heidelberg. ISBN: 978-3-662-52917-1. DOI: 10.1007/978-3-662-52919-5.
- Schlüss, D., C. Frey, and G. Ashcroft** (2016). “Consistent non-reflecting boundary conditions for both steady and unsteady flow simulations in turbomachinery applications”. In: *Proceedings of the VII European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS Congress 2016)*. Ed. by M. Papadrakakis. Athens: Institute of Structural Analysis and Antiseismic Research School of Civil Engineering National Technical University of Athens (NTUA) Greece, pp. 7403–7422. ISBN: 978-618-82844-0-1.
- Schmidt, O. T. and T. Colonius** (2020). “Guide to Spectral Proper Orthogonal Decomposition”. In: *AIAA Journal* 58.3, pp. 1023–1033. DOI: 10.2514/1.J058809.
- Schreiber, H.-A., W. Steinert, T. Sonoda, and T. Arima** (2004). “Advanced High-Turning Compressor Airfoils for Low Reynolds Number Condition—Part II: Experimental and Numerical Analysis”. In: *Journal of Turbomachinery* 126.4, pp. 482–492. ISSN: 0889-504X. DOI: 10.1115/1.1737781. URL: <https://asmedigitalcollection.asme.org/turbomachinery/article/126/4/482/447595/Advanced-High-Turning-Compressor-Airfoils-for-Low>.
- Schwarzbach, F., C. Müller-Schindewolffs, C. Bode, and F. Herbst** (2018). “The Effect of Turbulent Scales on Low-Pressure Turbine Aerodynamics: Part B — Scale Resolving Simulations”. In: *Proceedings of the ASME Turbo Expo: Turbomachinery Technical Conference and Exposition - 2018*. New York, N.Y.: The American Society of Mechanical Engineers. ISBN: 978-0-7918-5100-5. DOI: 10.1115/GT2018-75163.
- Scillitoe, A. D.** (2017). “Towards Predictive Eddy Resolving Simulations for Gas Turbine Compressors”. PhD thesis. Apollo - University of Cambridge Repository. DOI: 10.17863/CAM.16871.
- Sengupta, T. K., D. Sridar, S. Sarkar, and S. De** (2001). “Spectral analysis of flux vector splitting finite volume methods”. In: *International Journal for Numerical Methods in Fluids* 37.2, pp. 149–174. DOI: 10.1002/fld.168.
- Shu, C. W. and S. Osher** (1988). “Efficient implementation of essentially non-oscillatory shock-capturing schemes”. In: *Journal of Computational Physics* 77.2, pp. 439–471. ISSN: 0021-9991. DOI: 10.1016/0021-9991(88)90177-5.
- Shur, M. L., P. R. Spalart, M. K. Strelets, and A. K. Travin** (2014). “Synthetic Turbulence Generators for RANS-LES Interfaces in Zonal Simulations of Aerodynamic and Aeroacoustic Problems”. In: *Flow, Turbulence and Combustion* 93.1, pp. 63–92. ISSN: 1386-6184.

- Simoni, D., M. Ubaldi, P. Zunino, D. Lengani, and F. Bertini** (2012). “An Experimental Investigation of the Separated-Flow Transition Under High-Lift Turbine Blade Pressure Gradients”. In: *Flow, Turbulence and Combustion* 88.1-2, pp. 45–62. ISSN: 1573-1987. DOI: 10.1007/s10494-011-9375-7.
- Sirovich, L.** (1987). “Turbulence and the dynamics of coherent structures. I. Coherent structures”. In: *Quarterly of Applied Mathematics* 45.3, pp. 561–571. ISSN: 0033-569X. DOI: 10.1090/qam/910462.
- Sondergaard, R., R. B. Rivir, and J. P. Bons** (2002). “Control of Low-Pressure Turbine Separation Using Vortex-Generator Jets”. In: *Journal of Propulsion and Power* 18.4, pp. 889–895. DOI: 10.2514/2.6014.
- Souza, L. F., M. T. Mendonça, and M. A. F. Medeiros** (2005). “The advantages of using high-order finite differences schemes in laminar-turbulent transition studies”. In: *International Journal for Numerical Methods in Fluids* 48.5, pp. 565–582. DOI: 10.1002/flid.955.
- Spalart, P. R. and M. K. Strelets** (2000). “Mechanisms of transition and heat transfer in a separation bubble”. In: *Journal of Fluid Mechanics* 403, pp. 329–349. DOI: 10.1017/S0022112099007077.
- Spalart, P. R. and V. Venkatakrishnan** (2016). “On the role and challenges of CFD in the aerospace industry”. In: *The Aeronautical Journal* 120.1223, pp. 209–232. DOI: 10.1017/aer.2015.10.
- Spalart, P.** (2000). “Strategies for turbulence modelling and simulations”. In: *International Journal of Heat and Fluid Flow* 21.3, pp. 252–263. DOI: 10.1016/S0142-727X(00)00007-2.
- Spalart, P., W.-H. Jou, M. Strelets, and S. Allmaras** (1997). “Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach”. In: *Computer Science*, p. 11.
- Spencer, R., P. Przytarski, P. Adami, P. Grothe, and A. Wheeler** (2023). “Importance of Nonequilibrium Modeling for Compressors”. In: *Journal of Turbomachinery* 145.4. ISSN: 0889-504X. DOI: 10.1115/1.4054813.
- Steinert, W. and H. Starken** (1996). “Off-Design Transition and Separation Behavior of a CDA Cascade”. In: *Journal of Turbomachinery* 118.2, pp. 204–210. ISSN: 0889-504X. DOI: 10.1115/1.2836627.
- Suluksna, K., P. Dechaumphai, and E. Juntasaro** (2009). “Correlations for modeling transitional boundary layers under influences of freestream turbulence and pressure gradient”. In: *International Journal of Heat and Fluid Flow* 30.1, pp. 66–75. DOI: 10.1016/j.ijheatfluidflow.2008.09.004.

- Sutherland, W.** (1893). “LII. The viscosity of gases and molecular force”. In: *The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science* 36.223, pp. 507–531. ISSN: 1941-5982. DOI: 10.1080/14786449308620508.
- Taira, K., S. L. Brunton, S. T. M. Dawson, C. W. Rowley, T. Colonius, B. J. McKeon, O. T. Schmidt, S. Gordeyev, V. Theofilis, and L. S. Ukeiley** (2017). “Modal Analysis of Fluid Flows: An Overview”. In: *AIAA Journal* 55.12, pp. 4013–4041. DOI: 10.2514/1.J056060.
- Tennekes, H. and J. L. Lumley** (1999). *A first course in turbulence*. 17. print. Cambridge, Mass.: MIT Press. ISBN: 0262200198.
- Tollmien, W.** (1928). “Über die Entstehung der Turbulenz”. In: *Nachrichten von der Gesellschaft der Wissenschaften zu Göttingen* 1929, pp. 21–44. URL: <http://eudml.org/doc/59276>.
- Towne, A., O. T. Schmidt, and T. Colonius** (2018). “Spectral proper orthogonal decomposition and its relationship to dynamic mode decomposition and resolvent analysis”. In: *Journal of Fluid Mechanics* 847, pp. 821–867. DOI: 10.1017/jfm.2018.283.
- Tucker, P. G.** (2013). “Trends in turbomachinery turbulence treatments”. In: *Progress in Aerospace Sciences* 63, pp. 1–32. ISSN: 03760421. DOI: 10.1016/j.paerosci.2013.06.001.
- Tucker, P. G.** (2014). *Unsteady Computational Fluid Dynamics in Aeronautics*. Vol. 104. SpringerLink Bücher. Dordrecht: Springer. ISBN: 9789400770492. DOI: 10.1007/978-94-007-7049-2.
- Tyacke, J. C. and P. G. Tucker** (2015). “Future Use of Large Eddy Simulation in Aero-engines”. In: *Journal of Turbomachinery* 137.8. ISSN: 0889-504X. DOI: 10.1115/1.4029363.
- Tyacke, J., P. Tucker, R. Jefferson-Loveday, N. Rao Vadlamani, R. Watson, I. Naqavi, and X. Yang** (2014). “Large Eddy Simulation for Turbines: Methodologies, Cost and Future Outlooks”. In: *Journal of Turbomachinery* 136.6. ISSN: 0889-504X. DOI: 10.1115/1.4025589.
- Vadlamani, N. R.** (2014). “Numerical Investigation of Separated flows in Low Pressure Turbines”. Doctoral thesis. University of Cambridge.
- Van Leer, B.** (1979). “Towards the ultimate conservative difference scheme. V. A second-order sequel to Godunovs method”. In: *Journal of Computational Physics* 32.1. ISSN: 0021-9991. DOI: 10.1016/0021-9991(79)90145-1.
- Virtanen, P. et al.** (2020). “SciPy 1.0: fundamental algorithms for scientific computing in Python”. In: *Nature methods* 17.3, pp. 261–272. ISSN: 1548-7091. DOI: 10.1038/s41592-019-0686-2.

- Volino, R. J.** (2002a). “Separated Flow Transition Under Simulated Low-Pressure Turbine Airfoil Conditions—Part 1: Mean Flow and Turbulence Statistics”. In: *Journal of Turbomachinery* 124.4, pp. 645–655. ISSN: 0889-504X. DOI: 10.1115/1.1506938.
- Volino, R. J.** (2002b). “Separated Flow Transition Under Simulated Low-Pressure Turbine Airfoil Conditions—Part 2: Turbulence Spectra”. In: *Journal of Turbomachinery* 124.4, pp. 656–664. ISSN: 0889-504X. DOI: 10.1115/1.1506939.
- Volino, R. J. and L. S. Hultgren** (2001). “Measurements in Separated and Transitional Boundary Layers Under Low-Pressure Turbine Airfoil Conditions”. In: *Journal of Turbomachinery* 123.2, pp. 189–197. ISSN: 0889-504X. DOI: 10.1115/1.1350408.
- Vreman, B., B. Geurts, and H. Kuerten** (1995). “A priori tests of large eddy simulation of the compressible plane mixing layer”. In: *Journal of Engineering Mathematics* 29.4, pp. 299–327. ISSN: 0022-0833. DOI: 10.1007/BF00042759.
- Walker, G. J.** (1993). “The Role of Laminar-Turbulent Transition in Gas Turbine Engines: A Discussion”. In: *Journal of Turbomachinery* 115.2, pp. 207–216. ISSN: 0889-504X. DOI: 10.1115/1.2929223.
- Wallace, J. M., H. Eckelmann, and R. S. Brodkey** (1972). “The wall region in turbulent shear flow”. In: *Journal of Fluid Mechanics* 54.1, pp. 39–48. DOI: 10.1017/S0022112072000515.
- Walters, D. K. and D. Cokljat** (2008). “A Three-Equation Eddy-Viscosity Model for Reynolds-Averaged Navier–Stokes Simulations of Transitional Flow”. In: *Journal of Fluids Engineering* 130.12, p. 121401. ISSN: 0098-2202. DOI: 10.1115/1.2979230.
- Walters, D. K. and J. H. Leylek** (2004). “A New Model for Boundary Layer Transition Using a Single-Point RANS Approach”. In: *Journal of Turbomachinery* 126.1, pp. 193–202. ISSN: 0889-504X. DOI: 10.1115/1.1622709.
- Wang, R. and Z. Xiao** (2021). “Influence of free-stream turbulence on the aerodynamic performance of a three-dimensional airfoil”. In: *AIP Advances* 11.7. DOI: 10.1063/5.0054619.
- Weatheritt, J., R. Pichler, R. D. Sandberg, G. Laskowski, and V. Michelassi** (2017). “Machine Learning for Turbulence Model Development Using a High-Fidelity HPT Cascade Simulation”. In: *Proceedings of the ASME Turbo Expo: Turbine Technical Conference and Exposition - 2017*. New York, N.Y.: The American Society of Mechanical Engineers. ISBN: 978-0-7918-5079-4. DOI: 10.1115/GT2017-63497.
- Weber, A. and M. Sauer** (2016). *PyMesh - Template Documentation*. Technical report 146. DLR-IB-AT-KP-2016-146. Linder Hoehe, Cologne, Germany: German Aerospace Center (DLR), Institute of Propulsion Technology.

- Weiss, J.** (2019). “A Tutorial on the Proper Orthogonal Decomposition”. In: *AIAA Aviation 2019 Forum*. DOI: 10.14279/depositonce-8512. URL: <https://api.semanticscholar.org/CorpusID:199588490>.
- Wheeler, A., R. D. Sandberg, N. D. Sandham, R. Pichler, V. Michelassi, and G. Laskowski** (2016). “Direct Numerical Simulations of a High-Pressure Turbine Vane”. In: *Journal of Turbomachinery* 138.7. ISSN: 0889-504X. DOI: 10.1115/1.4032435.
- Wheeler, A. P. S., A. M. J. Dickens, and R. J. Miller** (2018). “The Effect of Nonequilibrium Boundary Layers on Compressor Performance”. In: *Journal of Turbomachinery* 140.10. ISSN: 0889-504X. DOI: 10.1115/1.4040094.
- Wilcox, D. C.** (1988). “Reassessment of the scale-determining equation for advanced turbulence models”. In: *AIAA Journal* 26.11, pp. 1299–1310. ISSN: 0001-1452.
- Winhart, B.** (2021). “Large-eddy simulation of a modified T106 low-pressure turbine stator under periodic wake impact”. Doctoral thesis. Ruhr-Universität Bochum, Universitätsbibliothek. DOI: 10.13154/294-8252.
- Winter, K.** (2016). “Einfluss einer Mehrfachwelligkeit auf die Aerodynamik eines sub- und transsonisch umströmten, ebenen Verdichtergitters”. Dissertation. Verlag Dr. Hut and RWTH Aachen.
- Wissink, J. G.** (2003). “DNS of separating, low Reynolds number flow in a turbine cascade with incoming wakes”. In: *International Journal of Heat and Fluid Flow* 24.4, pp. 626–635. DOI: 10.1016/S0142-727X(03)00056-0.
- Wissink, J. G. and W. Rodi** (2006). “Direct numerical simulation of flow and heat transfer in a turbine cascade with incoming wakes”. In: *Journal of Fluid Mechanics* 569, pp. 209–247. DOI: 10.1017/S002211200600262X.
- Wissink, J. and W. Rodi** (2004). “DNS of a Laminar Separation Bubble Affected by Free-Stream Disturbances”. In: *Direct and large-eddy simulation V*. Ed. by R. Friedrich, B. Geurts, and O. Métais. Vol. 9. ERCOFTAC Series. Dordrecht, Netherlands: Springer-Science+Business Media B.V, pp. 213–220. ISBN: 978-90-481-6575-9. DOI: 10.1007/978-1-4020-2313-2₂₃.
- Wu, X. and P. A. Durbin** (2001). “Evidence of longitudinal vortices evolved from distorted wakes in a turbine passage”. In: *Journal of Fluid Mechanics* 446, pp. 199–228. DOI: 10.1017/S0022112001005717.
- Wu, X., R. G. JACOBS, J. C. R. HUNT, and P. A. DURBIN** (1999). “Simulation of boundary layer transition induced by periodically passing wakes”. In: *Journal of Fluid Mechanics* 398, pp. 109–153. DOI: 10.1017/S0022112099006205.
- Xiong, Z. and S. K. Lele** (2007). “Stagnation-point flow under free-stream turbulence”. In: *Journal of Fluid Mechanics* 590, pp. 1–33. ISSN: 0022-1120.

- Yang, Z. and P. R. Voke** (2001). “Large-eddy simulation of boundary-layer separation and transition at a change of surface curvature”. In: *Journal of Fluid Mechanics* 439, pp. 305–33. DOI: 10.1017/S0022112001004633.
- Zaki, T., P. Durbin, J. Wissink, and W. Rodi** (2006). “Direct Numerical Simulation of By-Pass and Separation-Induced Transition in a Linear Compressor Cascade”. In: *Proceedings of the ASME Turbo Expo 2006*. New York, NY: ASME, pp. 1421–1429. ISBN: 0-7918-4241-X. DOI: 10.1115/GT2006-90885.
- Zaki, T. A., J. G. Wissink, P. A. Durbin, and W. Rodi** (2009). “Direct Computations of Boundary Layers Distorted by Migrating Wakes in a Linear Compressor Cascade”. In: *Flow, Turbulence and Combustion* 83.3, pp. 307–322. ISSN: 1573-1987. DOI: 10.1007/s10494-009-9216-0. URL: <https://link.springer.com/article/10.1007/s10494-009-9216-0>.
- Zaki, T. A., J. G. Wissink, W. Rodi, and P. A. Durbin** (2010). “Direct numerical simulations of transition in a compressor cascade: the influence of free-stream turbulence”. In: *Journal of Fluid Mechanics* 665, pp. 57–98. DOI: 10.1017/S0022112010003873.
- Zang, T. A., S. E. Krist, and M. Y. Hussaini** (1989). “Resolution requirements for numerical simulations of transition”. In: *Journal of Scientific Computing* 4.2, pp. 197–217. ISSN: 0885-7474. DOI: 10.1007/BF01061501.
- Zhao, Y., H. D. Akolekar, J. Weatheritt, V. Michelassi, and R. D. Sandberg** (2020a). “RANS turbulence model development using CFD-driven machine learning”. In: *Journal of Computational Physics* 411, p. 109413. ISSN: 0021-9991. DOI: 10.1016/j.jcp.2020.109413.
- Zhao, Y. and R. D. Sandberg** (2020b). “Using a New Entropy Loss Analysis to Assess the Accuracy of RANS Predictions of an High-Pressure Turbine Vane”. In: *Journal of Turbomachinery* 142.8. ISSN: 0889-504X. DOI: 10.1115/1.4046531. URL: <http://asmedigitalcollection.asme.org/turbomachinery/article/142/8/081008/1074991/Using-a-New-Entropy-Loss-Analysis-to-Assess-the>.
- Zhu, L., W. Zhang, J. Kou, and Y. Liu** (2019). “Machine learning methods for turbulence modeling in subsonic flows around airfoils”. In: *Physics of Fluids* 31.1. ISSN: 1070-6631. DOI: 10.1063/1.5061693.
- Zienkiewicz, O. C., R. L. Taylor, and P. Nithiarasu** (2014). *The finite element method for fluid dynamics*. Seventh edition. Oxford: Butterworth-Heinemann. ISBN: 9780080951379. URL: <http://www.sciencedirect.com/science/book/9781856176354>.
- Zou, Z., P. Xu, Y. Chen, L. Yao, and C. Fu** (2024). “Application of artificial intelligence in turbomachinery aerodynamics: progresses and challenges”. In: *Artificial Intelligence Review* 57.8, p. 222. ISSN: 1573-7462. DOI: 10.1007/s10462-024-10867-3. URL: <https://doi.org/10.1007/s10462-024-10867-3>.

Appendix

A Modal Analysis

Modal analysis, including Proper Orthogonal Decomposition (POD) and Spectral Proper Orthogonal Decomposition (SPOD), offers a robust framework for the analysis of flow losses in turbomachinery components. This analysis involves the decomposition of complex, unsteady flow fields into dominant coherent structures, facilitating the identification of energy-dominant flow features. Examples of such features include separation zones, wake structures and turbulence-induced losses. The extraction of spatial modes that capture the most significant kinetic energy content by POD facilitates the identification of these features.

In contrast, the SPOD method employs a frequency-domain analysis to identify frequency-dependent loss mechanisms, encompassing phenomena such as blade passing effects, vortex shedding, and acoustic interactions. By isolating the most significant modes, these methods facilitate a more profound comprehension of loss generation, enhance turbulence modeling, and guide aerodynamic design modifications to improve efficiency and mitigate performance penalties in turbomachinery applications.

In the following sections, a basic overview of the functional principles behind the decomposition methods, employed in this work, is presented.

A.1 Proper Orthogonal Decomposition (POD)

Even simple aerodynamic configurations can produce complex flow phenomena with a broad spectrum of temporal and spatial features. To analyze these flows, it is standard practice to first identify and extract key physical features—commonly referred to as modes—through modal decomposition of experimental or numerical data sets. For a comprehensive review of the various modal decomposition techniques relevant to fluid mechanics, see Taira et al. (2017).

In this work, a specific variant of the Proper Orthogonal Decomposition (POD) method, namely, the spectral proper orthogonal decomposition (SPOD) method, is employed for the modal analysis of unsteady three-dimensional data sets. Here, a brief overview of the basic functionalities of both methods is provided, while more detailed discussions can be found in Berkooz et al. (1993), Weiss (2019) and Taira et al. (2017).

The POD methods forms the foundation for SPOD. POD is a modal decomposition technique designed to extract the most energetic modes from a given flow variable. Originally introduced by Lumley (1979) as a statistical tool for identifying coherent structures in turbulent flow fields, the key idea behind POD is to decompose a fluctuating vector field,

$q'(x, t)$, into a series of deterministic spatial functions, $\phi_k(x)$, modulated by time-dependent coefficients, $a_k(t)$. This decomposition can be expressed as:

$$q'(x, t) = \sum_{k=1}^{\infty} a_k(t) \phi_k(x) \quad (\text{A.1})$$

where $x = (x, y, z)$ represents the position vector, and $q'(x, t) = q(x, t) - \bar{q}(x)$ is a fluctuating variable (such as a velocity) vector field.

When the POD is applied to velocity components $c = (c_1, c_2, c_3)^T$, where $q' = c' = c - \bar{c}$, represents the velocity fluctuations, each POD mode captures a distinct fraction of the total turbulent kinetic energy (TKE) of the flow. The POD method provides an optimal set of orthogonal basis functions, ensuring that a finite number of these modes retain more TKE than any other set of orthogonal modes.

In this work, the snapshot POD method introduced by Sirovich (1987) is employed, which offers a significant advantage over the classical (spatial) POD approach by drastically reducing storage requirements. This makes the POD method particularly well-suited for analyzing large datasets, such as those used in this study. The POD analysis is performed as follows:

1. The first step in the POD analysis is constructing the snapshot matrix U . For the three-dimensional dataset examined in this work, the resulting matrix has dimensions spatial resolution $n \times$ time instances m .

To create this matrix, the three-dimensional vector fields of the velocity fluctuation components are reorganized into one-dimensional line vectors. Each of these vectors is then horizontally concatenated and stacked, forming a structured matrix where each snapshot (i.e., a velocity field at a given time instance) corresponds to a single row in the snapshot matrix. This structure is illustrated as follows:

$$U = \begin{bmatrix} c'_1(x, t_1) & c'_2(x, t_1) & c'_3(x, t_1) \\ c'_1(x, t_2) & c'_2(x, t_2) & c'_3(x, t_2) \\ \vdots & \vdots & \vdots \\ c'_1(x, t_m) & c'_2(x, t_m) & c'_3(x, t_m) \end{bmatrix} \quad (\text{A.2})$$

Each row in U represents a snapshot of the flow at a particular time step, capturing the velocity fluctuations across all spatial points. This matrix serves as the foundation for further analysis using singular value decomposition (SVD) or other decomposition techniques to extract dominant flow structures.

2. In the next step the covariance matrix C is computed as following:

$$C = \frac{1}{m-1} U U^T \quad (\text{A.3})$$

The resulting matrix has the dimension $m \times m$

3. The POD modes from the Eigenvectors $\phi_k(x)$ of the covariance matrix C is determined as:

$$C \Phi_k(x) = \lambda_k \Phi_k(x) \quad (\text{A.4})$$

The Eigenvalues λ_k obtained from the POD analysis represent the amount of turbulent kinetic energy (TKE) captured by the respective POD modes. The eigenvalues are sorted in descending order, meaning that the first mode contains the largest portion of the TKE, followed by the second mode, and so on. This ordering allows for an efficient representation of the flow, where a small number of dominant modes can capture most of the flow's energy.

4. In the last step the temporal coefficients $a_k(t)$ are calculated

$$a_k(t) = \langle U^T, \Phi_k(x) \rangle \quad (\text{A.5})$$

In practical applications, only a small subset of the most energetic POD modes is often needed to reconstruct the dominant flow structures, significantly reducing the complexity of the dataset while preserving the essential dynamics of the flow.

A.2 Spectral Proper Orthogonal Decomposition (SPOD)

Similar to standard POD, SPOD identifies an optimal orthogonal basis for the data, where a subset of modes captures a larger portion of the total energy (or variance) than any other orthogonal basis. However, unlike traditional POD, which considers only spatial orthogonality, SPOD accounts for variations in both space and time, making it better suited for analyzing spatiotemporal coherence in flow data.

SPOD is a special case of space–time decomposition that relies on the assumption of statistical stationarity, meaning that the mean and variance remain constant over time. Under these conditions, the resulting modes are harmonic in time and are computed at each frequency from the data. In essence, SPOD combines the advantages of Dynamic Mode Decomposition (DMD)—which captures temporal correlations—with the optimal energy representation of POD. Additionally, SPOD modes are closely linked to resolvent analysis of forced, linear operators, providing further insights into turbulent flow modeling. While the POD aspect of SPOD follows a straightforward eigenvector decomposition (as described earlier), the spectral component is more complex. The core algorithm of SPOD is based on Welch's method, an averaging technique that consistently estimates the cross-spectral density of a time series. In classical POD, temporal correlations between extracted structures are lost, which led to the development of DMD Schmidt et al. (2020). Unlike POD, DMD approximates the eigenmodes of a linear operator that maps the state of a flow from one time step to the next, allowing it to capture flow structures that grow or decay over time. However, DMD modes are neither orthogonal nor optimal in representing flow statistics.

For statistically stationary flows, Towne et al. (2018) demonstrated that SPOD modes are equivalent to DMD modes (assuming the data are centered). In this context, SPOD can be seen as an optimal averaging of an ensemble of DMD modes computed from multiple realizations of the same flow. This optimality refers to SPOD's ability to best represent the second-order space–time statistics (covariance) of the flow.

A key step in SPOD is constructing the cross-spectral density (CSD) matrix, which captures the spatial correlations between different points in the flow at a given frequency. In practice, SPOD typically employs Welch's method to estimate the CSD matrix. This

approach enhances the accuracy of spectral estimation by dividing the time series into overlapping segments and averaging their Fourier transforms. As a result, Welch's method provides a more robust and reliable estimate of the CSD:

$$S(x, x', f) = \int_{-\infty}^{\infty} C(x, x', \tau) e^{-i2\pi f\tau} d\tau \quad (\text{A.6})$$

where $\tau = t - t'$, the cross-spectral density (CSD) matrix can be estimated from the data using Welch's method, a widely used technique in power spectral density (PSD) calculations. This method enhances the accuracy of spectral estimates by averaging over multiple data segments.

The following steps are preformed during the Welch method for SPOD:

1. The time series is divided into multiple overlapping segments.
2. A Fourier transform is applied to each segment.
3. Each segment contains a fixed number of snapshots, and the overlapping approach improves the spectral estimate's accuracy.

Welch's method is well-established, allowing for careful selection of parameters and providing a systematic way to assess the uncertainty of the estimation based on data quality. The discrete Fourier transform (DFT) of each realization and its inverse in time are then computed as follows:

$$\hat{q}^{(k)}(f_m) = \sum_{j=0}^{N_{\text{FFT}}-1} q^{(k)}(t_{j+1}) e^{-i2\pi jm/N_{\text{FFT}}}, \quad k = -N_{\text{FFT}}/2 + 1, \dots, N_{\text{FFT}}/2 \quad (\text{A.7})$$

$$q^{(k)}(t_{j+1}) = \frac{1}{N_{\text{FFT}}} \sum_{m=-N_{\text{FFT}}/2+1}^{N_{\text{FFT}}/2} \hat{q}^{(k)}(f_m) e^{i2\pi jm/N_{\text{FFT}}}, \quad j = 0, \dots, N_{\text{FFT}} - 1 \quad (\text{A.8})$$

Once transformed, the data are represented at discrete frequencies, $f_m = mN_{\text{FFT}}/T$, $m = -N_{\text{FFT}}/2, \dots, N_{\text{FFT}}$. After applying the Fourier transform to the ensemble of data, the remaining analysis is performed separately at each frequency. The data matrix is constructed as following

$$\hat{Q} = \begin{bmatrix} \left| \hat{q}^{(1)} \right\rangle & \left| \hat{q}^{(2)} \right\rangle & \dots & \left| \hat{q}^{(N)} \right\rangle \end{bmatrix}, \quad \hat{Q} \in \mathbb{C}^{M \times N} \quad (\text{A.9})$$

M is the total number of variables multiplied by number of spatial points, and N is the number of realizations (time instances). In this formulation, there is one $\hat{Q}$ per frequency, and the following steps are repeated for each frequency. The CSD, at each frequency, is calculated as following:

$$\hat{C} = \frac{1}{N-1} \hat{Q} \hat{Q}^H \quad (\text{A.10})$$

in the next step, the SPOD modes and eigenvalues are computed with the help of the snapshot method (as already described before).

In general, POD is well-suited for analyzing datasets that exhibit variations in time or space. To maximize its effectiveness, it is crucial to apply POD to dimensions where the most significant variations occur, such as time, space, or other relevant parameters.

B $k - \varepsilon$ turbulence model

The $k - \varepsilon$ model is one of the oldest and most widely used two-equation turbulence models. The standard form of the model, as commonly applied today, originates from the work of Jones et al. (1972), and was later refined by Launder et al. (1974). The model's development can be traced back to a series of scientific publications from the late 1960s and early 1970s (Launder et al. (1974), Pope (2000), Leschziner (2015)).

The $k - \varepsilon$ model is based on two transport equations, one for the turbulent kinetic energy k (TKE) and one for the dissipation rate ε :

$$\frac{D\rho k}{Dt} = P_k - D_k + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (\text{B.1})$$

$$\frac{D\rho \varepsilon}{Dt} = P_\varepsilon - D_\varepsilon + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\varepsilon \mu_t) \frac{\partial \varepsilon}{\partial x_j} \right] \quad (\text{B.2})$$

where the production and destruction terms are given by:

$$P_k = \bar{\tau}_{t,ij} \frac{\partial \bar{u}_i}{\partial x_j}, \quad D_k = \rho \varepsilon \quad (\text{B.3})$$

$$P_\varepsilon = C_1 \frac{\varepsilon}{k} \bar{\tau}_{t,ij} \frac{\partial \bar{c}_i}{\partial x_j}, \quad D_\varepsilon = C_2 \frac{\rho \varepsilon^2}{k} \quad (\text{B.4})$$

The system of equations is closed using the turbulent viscosity formulation:

$$\mu_t = C_\mu \rho k^2 / \varepsilon \quad (\text{B.5})$$

In the above equations, C represents the model's calibration parameters, while σ governs the turbulent diffusion of the respective transport variable. As noted by Pope (2000), the $k - \varepsilon$ model is widely used and well suited for simple flow scenarios. However, it has significant limitations in predicting complex three-dimensional flows, strong curvature effects, and circular jets. While specific recalibrations exist for specialized cases, they often come at the expense of the model's general reliability.

C $\gamma - \text{Re}_\theta$ transition model

Traditional turbulence models often struggle to accurately predict boundary layer transition and tend to significantly overestimate the onset of turbulence, as noted by Langtry (2006). To address this limitation, incorporating transition models has become standard practice in computational fluid dynamics. These models introduce an intermittency factor, γ , which represents the fraction of time the boundary layer remains in a turbulent

state. By modifying the production term of turbulent kinetic energy, γ ensures that the boundary layer remains laminar when $\gamma = 0$.

Various transition modeling approaches have been developed over the years, including analytical, empirical, and physics-based methods. This thesis focuses on the γ - Re_θ model, which will be discussed in detail.

The γ - Re_θ was first introduced by Menter et al. (2004) and later refined by Langtry (2006). A comprehensive set of correlations was subsequently published Langtry et al. (2009). In the mean time, the research community developed alternative formulations, such as those by Krause et al. (2008), Malan et al. (2012), and Suluksna et al. (2009). Further refinements were made even after the model's official publication, such as Fehrs (2018) recalibration, which improved transition predictions for low-turbulence aerodynamic configurations. Fehrs' approach, like Menter et al. (2015), omits the $\text{Re}_{\theta t}$ transport equation and introduces fundamental changes to the model's correlations. One of the key advantages of the γ - Re_θ model, as formulated by Langtry et al. (2009), is its fully local implementation. Unlike older approaches that require flow property integration across the boundary layer thickness, this model allows for straightforward application to unstructured grids and efficient parallel computing. The model employs two transport equations for γ and $\text{Re}_{\theta t}$, which track boundary layer history. Transition is triggered when the local Reynolds number, based on shear rate, surpasses a critical threshold.

$$\text{Re}_v = \frac{\rho y^2}{\mu} S \quad (\text{C.1})$$

where ρ is density, y is the distance from the wall, S is the shear rate and μ is the dynamic viscosity. For a laminar boundary layer without pressure gradients,

$$\text{Re}_\theta = \frac{\max(\text{Re}_v)}{2.193} \quad (\text{C.2})$$

When pressure gradients are moderate ($2.3 < H_{12} < 2.9$), the error is under 10% (Menter et al. (2004)).

The critical Reynolds number is determined based on the correlations from Abu-Ghannam et al. (1980) and considers turbulence intensity and acceleration parameters λ_θ .

$$\text{Re}_{\theta t} = f(TI, \lambda_\theta) \quad (\text{C.3})$$

$$\lambda_\theta = \frac{\theta^2}{\nu} \frac{dC}{ds} \quad (\text{C.4})$$

where θ is the momentum thickness, ν is the kinematic viscosity, and C is the free-stream velocity. The model transports $\text{Re}_{\theta t}$ into the boundary layer using convection and diffusion and adjusts the local Reynolds number to predict transition accurately.

$$\frac{\partial (\rho \tilde{\text{Re}}_{\theta t})}{\partial t} + \frac{\partial (\rho C_j \tilde{\text{Re}}_{\theta t})}{\partial x_j} = P_{\theta t} + \frac{\partial}{\partial x_j} \left[\sigma_{\theta t} (\mu + \mu_t) \frac{\partial \tilde{\text{Re}}_{\theta t}}{\partial x_j} \right] \quad (\text{C.5})$$

where $P_{\theta t}$ is the production term and $\sigma_{\theta t}$ is a model constant. The production term is given by

$$P_{\theta t} = c_{\theta t} \frac{\rho}{t} (\text{Re}_{\theta t} - \tilde{\text{Re}}_{\theta t}) (1.0 - F_{\theta t}) \quad (\text{C.6})$$

with $c_{\theta t}$ as a calibration constant and $F_{\theta t}$ as a switching function separating boundary layer and free-stream flow. In addition to modeling attached boundary layers, the γ - Re_θ model can handle separation-induced transition. In such cases, a value for γ_{sep} is calculated to account for shear layer behavior in separation bubbles.

$$\gamma_{\text{sep}} = \min \left(s_1 \max \left[0, \left(\frac{\text{Re}_v}{3.235 \text{Re}_{\theta c}} \right) - 1 \right] F_{\text{reattach}}, 2 \right) F_{\theta t} \quad (\text{C.7})$$

where γ_{sep} accounts for shear layers in separation bubbles, and F_{reattach} controls how fast the bubble reattaches.

Finally, the γ equation for the intermittency factor is:

$$\frac{\partial(\rho\gamma)}{\partial t} + \frac{\partial(\rho C_j \gamma)}{\partial x_j} = P_\gamma - D_\gamma + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_f} \right) \frac{\partial \gamma}{\partial x_j} \right] \quad (\text{C.8})$$

where the production term P_γ is:

$$P_\gamma = F_{\text{length}} c_{a1} \rho S [\gamma F_{\text{onset}}]^{0.5} (1 - c_{e1} \gamma) \quad (\text{C.9})$$

and the destruction term D_γ is:

$$D_\gamma = c_{a2} \rho \Omega \gamma F_{\text{turb}} (c_{e2} \gamma - 1) \quad (\text{C.10})$$

where F_{turb} is a function of the ratio of turbulent viscosity to dynamic viscosity.

This approach can lead to up to twice the production of turbulent kinetic energy in shear layers. Overall, the model demonstrates high adaptability to different flow conditions, including transition triggered by surface roughness, vortex-generating jets, and separation-induced effects. Its modular structure, combined with the use of local parameters, enhances its flexibility for a wide range of aerodynamic applications.

D Error analysis

The following derivation of mean-square error and confidence interval are taken from Bergmann 2023 and Tennekes et al. 1999. The mean-square error of a variable ϕ is estimated as

$$e_N^2(\bar{\phi}) \approx \frac{D_N(\phi) \text{Var}_N(\phi)}{N} \quad (\text{D.1})$$

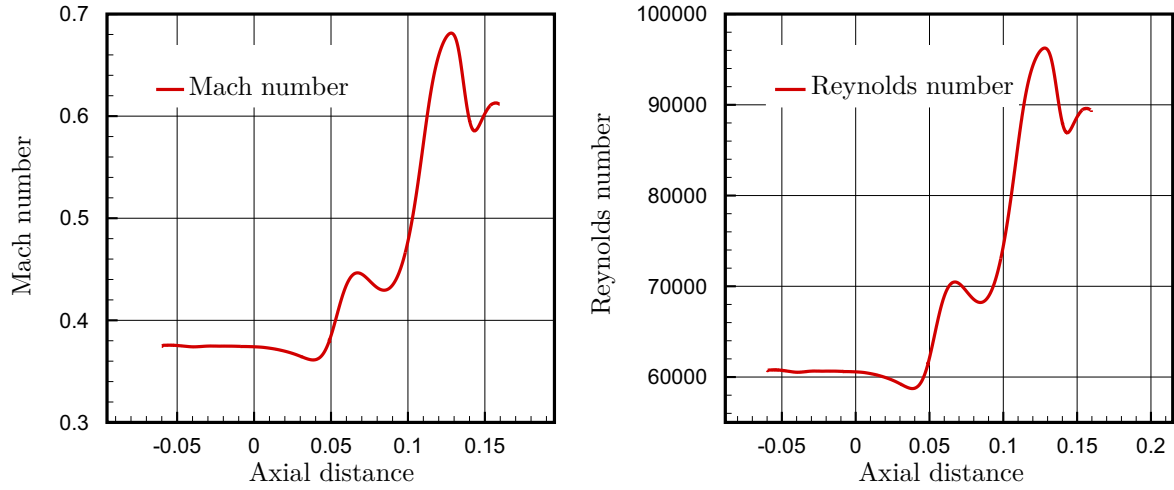


Figure D.1: Axial evolution of Mach and Reynolds number along a streamline in the middle of the passage

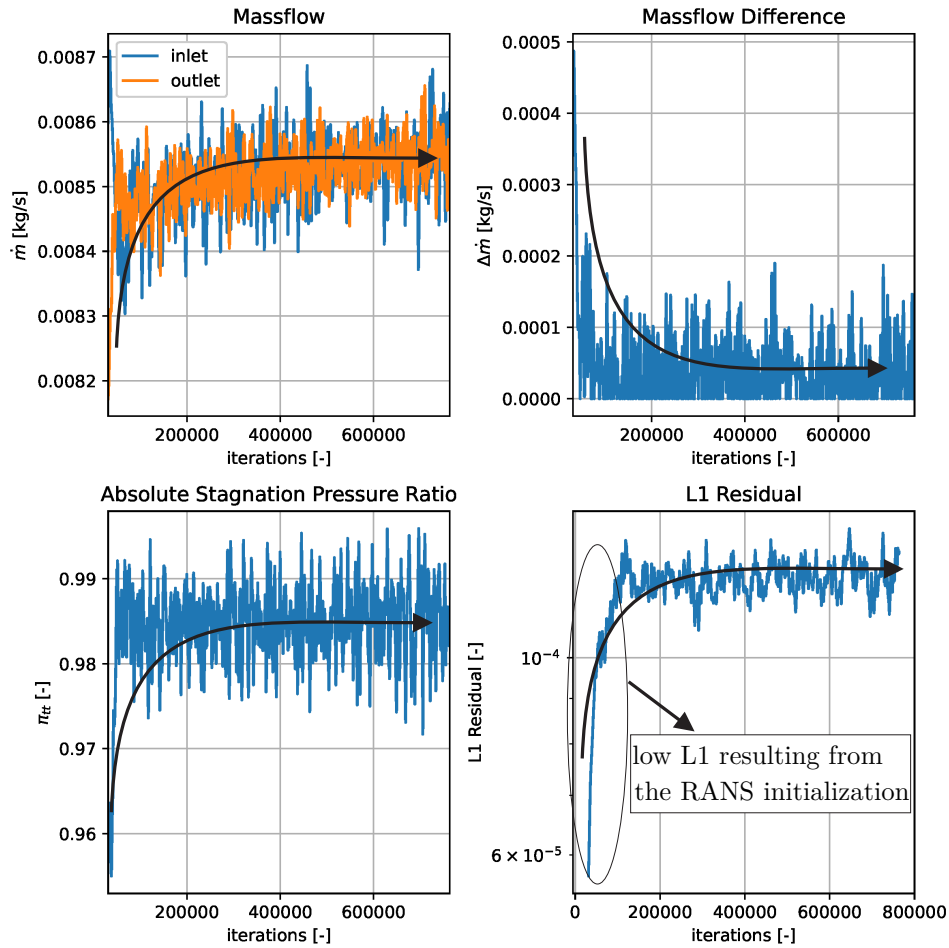


Figure D.2: Monitoring the convergence of massflow, total pressure ratio and L1

where $D_N(\phi)$ is the minimal number of samples between two independent observations given by

$$D_N(\phi) \approx 1 + 2 \sum_{\tau=1}^{\text{until } \rho_{N,\tau} < 0} \rho_{N,\tau}(\phi) \quad (\text{D.2})$$

Here, $\rho_{N,\tau}$ is the straight-forward estimate of the autocorrelation with lag τ

$$\rho_{N,\tau}(\phi) = \frac{\frac{1}{N-\tau} \sum_{i=\tau}^{N-1} (\phi_i - \bar{\phi}) (\phi_{i-\tau} - \bar{\phi})}{\text{Var}_N(\phi)} \quad (\text{D.3})$$

Based on the outcome of the central limit theorem, one can define confidence intervals of a statistical moment with the estimated error following

$$\text{CI}_{68\%} \approx [\bar{\phi} - 1e_N; \bar{\phi} + 1e_N] \quad (\text{D.4})$$

$$\text{CI}_{95\%} \approx [\bar{\phi} - 1.96e_N; \bar{\phi} + 1.96e_N] \quad (\text{D.5})$$

Figure D.3 shows the convergence of total pressure loss in the wake region for different averaging duration (10, 20 and 30 through flows). From Fig. D.3, it is visible that the total pressure loss in wake is converged after 20 through flows (TF), or convective time units and the difference in peak loss between 10 TF and 20 TF averaging is about 2 %. The convective time unit is defined as chord length divided either by the outlet or inlet velocity. Figure D.3 shows also the confidence interval for the total pressure in the wake region for 20TF. This figure shows the deviations around the mean value of the total pressure after 20 TF averaging.

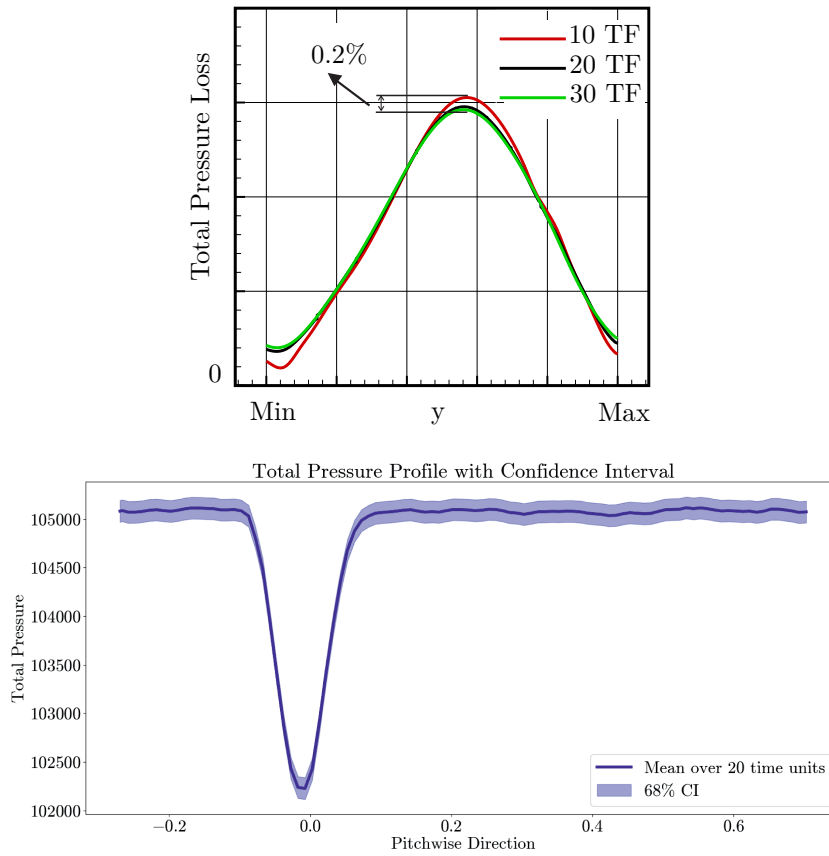


Figure D.3: Monitoring the convergence of averaging duration for total pressure wake loss (top) and total pressure in wake area (bottom). The corresponding 68 % confidence interval is shown, computed following Bergmann et al. (2022)