Robust aero-thermal design of high pressure turbines at uncertain exit conditions of low-emission combustion systems

Robuste aero-thermale Auslegung von Hochdruckturbinen bei unsicheren Austrittsbedingungen emissionsarmer Verbrennungssysteme
Zur Erlangung des akademischen Grades Doktor-Ingenieur (Dr.-Ing.)
genehmigte Dissertation von Marius Schneider aus Gießen
Tag der Einreichung: 15.04.2019
Tag der Prüfung: 17.07.2019
Darmstadt — D 17

1. Gutachten: Prof. Dr.-Ing. Heinz-Peter Schiffer
2. Gutachten: Prof. Dr.-Ing. Christian Hasse
Robust aero-thermal design of high pressure turbines at uncertain exit conditions of low-emission combustion systems
Robuste aero-thermale Auslegung von Hochdruckturbinen bei unsicheren Austrittsbedingungen emissionsarmer Verbrennungssysteme

Genehmigte Dissertation von Marius Schneider aus Gießen

1. Gutachten: Prof. Dr.-Ing. Heinz-Peter Schiffer
2. Gutachten: Prof. Dr.-Ing. Christian Hasse

Tag der Einreichung: 15.04.2019
Tag der Prüfung: 17.07.2019

Darmstadt — D 17

Bitte zitieren Sie dieses Dokument als:
URN: urn:nbn:de:tuda-tuprints-91124
URL: https://tuprints.ulb.tu-darmstadt.de/id/eprint/9112

Dieses Dokument wird bereitgestellt von tuprints, E-Publishing-Service der TU Darmstadt
http://tuprints.ulb.tu-darmstadt.de
tuprints@ulb.tu-darmstadt.de

Die Veröffentlichung steht unter folgender Creative Commons Lizenz:
Namensnennung – Keine kommerzielle Nutzung – Keine Bearbeitung 4.0 International
https://creativecommons.org/licenses/by-nc-nd/4.0/
Editor’s Preface

The series Research Reports from the Institute of Gas Turbines and Aerospace Propulsion accounts for the advances made in turbomachinery research and development at Technische Universität Darmstadt. Because of the strong application oriented focus of the research in this area, the academic problems reflect actual industrial development trends.

The current development foci adapt to the changing political, economic and ecological framework which keeps carrying the turbomachine towards the border of technological feasibility. In consequence, it is not unusual for findings to be transferred to the industrial application directly.

It is within this environment, that the industry and application oriented research works of this series originate. The reports describe current findings of experimental investigations and numerical simulations which were obtained at the Institute of Gas Turbines and Aerospace Propulsion at Technische Universität Darmstadt.

Heinz-Peter Schiffer
Darmstadt, 2019
Author’s Preface

Acknowledgements
First and foremost I would like to thank my supervisor Prof. Dr.-Ing. Heinz-Peter Schiffer for his guidance and support during the last five years. His overview, technical background and understanding of the industry have contributed greatly to this work. The Institute of Gas Turbines and Aerospace Propulsion has always been a creative and motivating working environment. Furthermore, I thank Prof. Dr.-Ing. Christian Hasse for the co-examination of the thesis and Max Jüngst, Manuel Wilhelm, Johannes Ratz and Jonathan Gründler for proofreading.

My work at GLR was supported by all colleagues on a daily basis. Specifically, I want to thank Barbara Löhr for handling the administration of the project and the finances, Gregor Schmid for applying for the research project, the LSTR team, especially Holger Werschnik, Manuel Wilhelm and Tom Ostrowski, for sharing their data and experience, the CFD team, especially Jonathan Hilgert and Faramarz Bakhtiari for the numerous on- and off-topic discussions, my fellow commuters Max Jüngst, Thomas König, Daniel Möller, Steffen Hormel, Felix Holzinger, Johannes Weinkauff and Christian Vey for wasting our time together and Christoph Brandstetter for the stimulating input and phases of distraction.

Many students have directly or indirectly supported this work with their final theses or projects. In particular, I want to thank Lars Ohde [132], Jonathan Gründler [60], Dimitri Ivanov [84], Hendrik Bäumler [13] and Andres Mejia-Wille [116].

The help of the industrial partner Rolls-Royce Deutschland is appreciated. In particular I would like to thank Roland Wilhelm, Knut Lehmann, Jerrit Dähnert and Marcus Meyer from the turbines aero-thermal group and Max Stauf er, Ruud Eggels, Thorsten Voigt and Sebastian Bake from the combustion group. The work of the engineering graduates Tobias Hahn and Pelayo Espinosa is also gratefully acknowledged.

Also, I would like to thank Michael Hanna of Turbo Science GmbH for sharing his experience and the support in solving numerous problems.

Last but not least, I want to thank my family, my friends and you, Magda, for tolerating my physical and mental absence during the last years.

Marius Schneider
Frankfurt am Main, 2019
Research Context and Funding

This thesis is embedded into a series of experimental and numerical investigations at the Rolls-Royce University Technology Centre “Combustor and Turbine Aero-thermal Interaction” at TU Darmstadt. The dissertation results to a large part from work that was funded by a scholarship of the Graduate School GRK 1344 “Instationäre Systemmodellierung von Flugtriebwerken” of Deutsche Forschungsgemeinschaft and the Luftfahrtforschungsprogramm LuFo V-2 “Advanced Components for Turbines” (AdCoTurb) of the Bundesministerium für Wirtschaft und Energie under grant FKZ 20T1312A. Calculations on the Lichtenberg high-performance computer of TU Darmstadt were conducted for this research.
Abstract

A key challenge in the development of novel, low-emission combustion systems in jet engines is the analysis of combustor turbine interaction. The exit conditions of the combustor are accounted for in the design of the first high pressure turbine stage in order to increase the efficiency of the system. Due to the extreme temperatures in jet engine combustors the knowledge of these conditions is subject to large uncertainties. The goal of this work is the development of a method to account for these uncertainties in design. This shall enable the development of robust components that do not fail if conditions deviate from the design point.

A major component of the method is a model that generates two-dimensional flow profiles of modern lean burn combustors based on a parameter set. These are used as boundary condition of a three dimensional flow simulation of the turbine. Stochastic deviations of the input parameters within the uncertainties can thus be accounted for. The developed process chain which couples parameters of turbine inlet conditions with performance parameters of the engine is analysed by means of statistical methods for uncertainty quantification. The model is able to reproduce both, conditions of a test rig as well as those in real engines, with sufficient accuracy.

Strong swirl at the combustor exit, which is characteristic for modern combustors, interacts with the first row of stator vanes of the turbine. Secondary flows in the vane passage, known from the literature, are influenced and additional structures are induced by the inlet swirl. By means of the developed process, a significant correlation between the circumferential position of the inlet swirl core and the radial position of the induced structures is identified. The relation transforms variations in the circumferential position of inlet swirl to variations in the local thermal load of the vanes and hub end wall and thus of the turbine’s life time. Uncertainties in thermal efficiency result mainly from uncertainties in the position of hot streaks at turbine inlet.
Contents

1 Introduction
   1.1 Background and Motivation .......................... 1
      1.1.1 Aircraft Emissions and Ecological Impact .......... 1
      1.1.2 Technical Measures to Reduce Emissions .......... 2
      1.1.3 Impact on the High Pressure Turbine .............. 4
      1.1.4 Analysis of Combustor Turbine Interaction .......... 5
      1.1.5 Uncertainty of Turbine Inflow Conditions .......... 7
   1.2 Placement of this Work, Scientific Approach and Research Objectives .... 8
   1.3 Thesis Outline ........................................ 10

2 State of the Art ........................................... 13
   2.1 Combustion and Emissions ............................. 13
      2.1.1 Geometry and Working Principle of Modern Combustion Chambers .......... 13
      2.1.2 Combustion Instabilities ........................... 14
      2.1.3 Combustor Exit Flow Field .......................... 16
   2.2 Turbine Aerodynamics .................................. 18
      2.2.1 Work Extraction in Turbines ....................... 18
      2.2.2 Efficiency and Losses .............................. 20
      2.2.3 Turbine Secondary Flows ............................ 22
      2.2.4 Turbine Capacity .................................... 23
   2.3 Turbine Heat Transfer and Cooling ...................... 24
      2.3.1 Heat Transfer ....................................... 24
      2.3.2 Film Cooling ....................................... 26
   2.4 Combustor Turbine Interaction .......................... 27
      2.4.1 Temperature Induced Secondary Flows ............... 27
      2.4.2 Experimental and Numerical Studies ............... 29
      2.4.3 Implications for this Work .......................... 37

3 Methods Review – CFD and UQ .............................. 39
   3.1 Fundamentals of Computational Fluid Dynamics .......... 39
      3.1.1 Conservation Equations .............................. 39
      3.1.2 Turbulent Dynamics and Scales ....................... 40
      3.1.3 Reynolds-Averaged Navier-Stokes Turbulence Models .......... 41
      3.1.4 Discretisation and Solution of Conservation Equations .......... 46
3.1.5 Numerical Error .................................................. 49
3.2 Aspects of Simulating Flow through High Pressure Turbines ...... 49
3.3 Numerical Analysis of Combustor Turbine Interaction ............ 51
  3.3.1 Shortcomings of Decoupling Combustor and Turbine ......... 51
  3.3.2 Discontinuity at the Combustor Turbine Interface ............ 51
  3.3.3 Scaling Turbine Inlet Conditions ............................ 53
3.4 Evaluation of the Flow Solver ...................................... 54
  3.4.1 Test Case, Mesh and Numerical Set-up ....................... 54
  3.4.2 Axial Inflow .................................................. 57
  3.4.3 Swirling Inflow .............................................. 62
  3.4.4 Summary of Solver Evaluation and Impact on further Studies 67
3.5 Uncertainty Quantification Methods ................................ 68
  3.5.1 Review of Fundamental Statistical Properties and Definitions 68
  3.5.2 Classification of Uncertainty in Technical Systems .......... 70
  3.5.3 Uncertainty Quantification ................................... 70
  3.5.4 Monte Carlo Sampling ........................................ 71
  3.5.5 Generalised Polynomial Chaos Expansions .................... 73
4  Method Development – Combustor Exit Flow Model ................. 77
  4.1 Introduction ....................................................... 77
    4.1.1 Overall Modelling Approach ................................. 77
    4.1.2 Characteristic Features of Lean Burn Traverses ............. 78
  4.2 Overall Parameterisation Scope and Strategy ...................... 79
    4.2.1 Structure, Inputs and Outputs ............................... 79
    4.2.2 Constraints .................................................. 80
  4.3 Modelling of In-Plane Velocity Distribution ...................... 80
    4.3.1 Swirl Field .................................................. 80
    4.3.2 Establishing Wall-Tangential Flow ......................... 82
    4.3.3 Modelling Entrainment of Surrounding Flow ............... 84
    4.3.4 Assembling the final In-Plane Vector Field ............... 85
  4.4 Modelling of Axial Velocity Distribution ........................ 85
  4.5 Modelling of Total Pressure Distribution ........................ 86
  4.6 Error Feedback in the Correction Loops of Axial Velocity and Pressure 87
  4.7 Modelling of Total Temperature Distribution ........................ 87
    4.7.1 Background Profile ......................................... 88
    4.7.2 Hot Spots .................................................... 88
    4.7.3 Accounting for Combustor Wall Film Cooling ............... 89
    4.7.4 Temperature Scaling ........................................ 89
  4.8 Modelling of Turbulence Distribution ................................ 89
  4.9 Summary of Model Parameters and Discussion of their Interdependence 90
B Appendix to Methods Review – CFD and UQ

B.1 Solving the Discretised Equations
  B.1.1 Properties of the Flow Solver
  B.1.2 Solution of the Coupled System of Equations
  B.1.3 Multigrid Algorithm

B.2 Modelling Transient Blade Row Interaction
  B.2.1 Mixing Planes
  B.2.2 Transient Blade Row Models

B.3 Numerical Simulation of Combustor Turbine Interaction
  B.3.1 Conserving Enthalpy Inflow instead of Turbine Inlet Temperature
  B.3.2 Variance of Enthalpy Inflow

B.4 Sampling Techniques
  B.4.1 Hammersley Sampling
  B.4.2 Latin Hypercube Sampling

B.5 Polynomial Chaos Expansions - Quadrature Methods

B.6 Verification of Uncertainty Quantification Methods
  B.6.1 Cycle Properties of Example Problem

B.7 Post-processing of Polynomial Chaos Analyses

C Appendix to Methods Development – Combustor Exit Flow Model

C.1 Modelling of the Velocity Field
  C.1.1 Lamb-Oseen Vortex
  C.1.2 Swirl Number Definition
  C.1.3 Modelling of Cross Flow from Converging End Walls
  C.1.4 Modelling of Flow Entrainment

C.2 Modelling of the Pressure Field
  C.2.1 Effects of Static Pressure and Mach Number on Total Pressure Variation
  C.2.2 Modelling of the 2D Pressure Profile

C.3 Blending of the Temperature Profile

C.4 Linking of First Order and Second Order Parameters

C.5 Evaluation of Combustor Exit Flow Model
  C.5.1 Matlab Genetic Algorithm
  C.5.2 Parameters varied in the Matching
  C.5.3 RRD-HPT – Computational Set-up and Mesh Study
  C.5.4 RRD-HPT – Circumferential Averages
  C.5.5 Matched Traverses
  C.5.6 RRD-HPT – Additional Test Cases of Matched Inlet Conditions
  C.5.7 RRD-HPT – Match of Wall Temperatures and Heat Transfer
  C.5.8 RRD-HPT – Inlet Velocity Correction
D Appendix to Results 185

D.1 RRD-HPT – Losses and Vane Secondary Flows Induced by Inlet Swirl 185
  D.1.1 RRD-HPT – Efficiency Variation at Pressure Outlet Condition . 185
  D.1.2 RRD-HPT – Comparison of SST and RSM Turbulence Modelling185
  D.1.3 Influence of Transient Blade Row Interaction ............... 187
  D.1.4 RRD-HPT – Efficiency Loss Correlation .................. 188
D.2 LSTR – Set-up of Swirler Computations .......................... 189
D.3 LSTR – Influence of Total Pressure Profile ..................... 190
D.4 LSTR – Correlations with Heat Transfer .......................... 191
D.5 E3E – Influence of Turbulence Modelling ........................ 192
  D.5.1 E3E – Influence on Swirl-Induced Secondary Flows ........ 192
  D.5.2 E3E – Influence on NGV1 Wall Temperatures .............. 192
D.6 E3E – Nusselt Number for Datum and Matched Inlet Conditions . 193
D.7 Deriving Uncertainties of the Hot Streak Position from Test Data . 194
D.8 E3E – Results of Uncertainty Quantification Studies ............ 195
  D.8.1 E3E – Uncertainty Propagation ............................... 195
  D.8.2 E3E – Correlation Matrices ................................. 195
  D.8.3 E3E – Relation of Film Cooling Effectiveness and Wall Temperature .................... 196
  D.8.4 E3E – Temperature-Weighted Standard Deviation of Wall Temperature ............. 197
  D.8.5 E3E – Vane Wall Temperature Correlation Coefficients .... 198
  D.8.6 E3E – Vane Nusselt Number Correlation Coefficients .... 199

E Appendix to Conclusion 201

E.1 Global Impact of Uncertainty in High Pressure Turbine Efficiency . . 201

List of Figures 203

List of Tables 207

List of Symbols 209

Bibliography 219
1 Introduction

1.1 Background and Motivation

1.1.1 Aircraft Emissions and Ecological Impact

The fight against climate change is one of the major challenges for mankind in the 21st century. The Intergovernmental Panel on Climate Change (IPCC) recently acknowledged a human-induced global warming of 1 °C compared to pre-industrial levels and pointed out substantial threats to the global ecosystem associated with further increases, caused mainly by the emission of carbon dioxide $CO_2$ (IPCC [83]). About 5% of anthropogenic radiative forcing$^1$ is caused by civil aviation (Lee et al. [111]). Aviation is a strongly growing sector and in 2008 the International Civil Aviation Organisation (ICAO) predicted the number of aircraft for civil transportation to increase by 4% annually (ICAO [77]).

The generation of thrust in aircraft engines is based on heat release from the combustion of kerosene with ambient air,

\[
C_nH_m + O_2 + N_2 + S \rightarrow \underbrace{CO_2 + H_2O}_\text{ideal combustion} + \underbrace{N_2 + O_2 + CO + SO_x + NO_x + UHC + soot}_\text{real combustion}
\] (1.1)

that generates the pollutants$^2$ soot, carbon oxide CO, unburned hydrocarbons (UHCs) and nitrogen oxides NO$_x$ (Masiol and Harrison [114]).

The emitted substances affect the environment in various ways. The greenhouse gases CO$_2$ and H$_2$O accumulating in the stratosphere reflect infrared radiation back to earth thus contributing to global warming (Farokhi [48]). Nitrogen oxide NO, emitted by subsonic aviation in the lower atmosphere, mainly acts as a catalyst for the production of ozone O$_3$, which also contributes to the greenhouse effect. NO$_2$ is mostly emitted at the ground at low-power conditions (Wormhoudt et al. [187]). It does not strongly impact climate change but directly affects human health as it is known to be responsible for cardiovascular and respiratory diseases (Masiol and Harrison [114]).

---

$^1$ Radiative forcing is a measure approximately proportional to the increase in global mean surface temperature.

$^2$ The products CO$_2$ and water vapor, H$_2$O, are inevitably generated during a complete combustion. Emitted sulfur oxides, SO$_x$, contribute to the pollution of the environment as well, but their formation is not governed by the combustion itself but rather by the fuel refinery process (Farokhi [48]). These substances do harm the environment, but are typically not included in the list of jet engine emissions.
The limits on emissions allowing a licensing of new engines are defined by ICAO’s standards on soot, NO\textsubscript{x}, UHC and CO (ICAO [78]) during landing-takeoff cycles. These are carried out at ground, simulating the engine’s emissions in the proximity of airports at altitudes below 3000 ft (0.9 km) where pollutants mix with the atmosphere. Emissions at cruise altitude (8 - 12 km) are currently not part of the licensing process (Masiol and Harrison [114]). Due to their detrimental effects on climate and health, in its 2017 Update to the Strategic Research & Innovation Agenda, the Advisory Council for Aviation Research and Innovation in Europe (ACARE) targets the reduction in CO\textsubscript{2} and NO\textsubscript{x} emissions of 2000 by 75 % and 90 %, respectively, by the year 2050. This poses a major challenge to engine manufacturers.

1.1.2 Technical Measures to Reduce Emissions

About 75 % of in-use aircraft are equipped with turbofan engines (Masiol and Harrison [114]). The architecture of such an engine is illustrated in Fig. 1.1 a). Downstream of the fan the air flow to the engine is divided into a stream \(\dot{m}_{\text{core}}\) through the core engine and a bypass stream \(\dot{m}_{\text{bypass}}\). In modern turbofan engines with bypass ratios \(\alpha := \dot{m}_{\text{bypass}}/\dot{m}_{\text{core}}\) exceeding 10 : 1 thrust is generated mostly by the fan. The core engine operates as gas turbine generating power for the fan, described by the idealised Brayton cycle (red in Fig. 1.1 b) with isentropic compression and expansion and isobaric heat transfer. The fan and compressor are usually mounted to different shafts (spools) with different speeds of rotation \(\omega\) in order to minimise aerodynamic loss and generation of noise at the fan blade tips. The power required to drive the high and low pressure compressor (HPC and LPC) and fan is extracted from the flow through the high and low pressure turbine (HPT and LPT), respectively.

![Components of a modern turbofan engine](image1.png)

![Brayton cycle](image2.png)

**Figure 1.1:** Turbofan components and cycle process (adapted from Rick [145])
Emissions are generated in an engine’s combustion chamber. The non-ideal products in Eqn. (1.1) are produced by different mechanisms which depend on the operating condition of the combustor. Whereas CO and UHCs result primarily from poor vaporisation or incomplete combustion at low temperatures and pressures (Farokhi [48]), the generation of thermal NO\textsubscript{x} requires the dissociation of oxygen at high temperatures above 1800 K as described by the extended Zeldovich mechanism (Lavoie et al. [109]). The concept of staged combustion has proven to be effective in reducing CO and UHC levels but fails to sufficiently reduce NO\textsubscript{x} formation (Farokhi [48]).

Modern concepts aim at the reduction of thermal NO\textsubscript{x} by avoiding high combustion temperatures. The rich burn-quick quench-lean burn (RQL) technology was first proposed in 1980 by Mosier and Pierce [125]. The basic idea is to achieve a fast transition of a rich towards a lean mixture by a rapid mixing with cool air (quenching). RQL can be regarded as the industrial standard in current civil jet engine combustors. A competing technology is the combustion of premixed fuel and air entirely in the lean regime. Thus, NO\textsubscript{x} emissions can be drastically reduced during take-off and climb (Lazik et al. [110]). Lean burn, however, still faces major challenges in combustion stability which is, why it is applied by only few in-service engines\(^3\). In his list of “remaining hot gas path challenges” Bunker [26] rates low emission combustor-turbine systems as a technology which has high potential impact but also poses highest technological risks.

Apart from directly inhibiting the generation of pollutants in the burner, a reduction of net emissions can be realised by increasing the engine’s overall efficiency \(\eta_o\), i.e., generating more thrust per emitted mole of pollutants. The overall efficiency is the product of a thermal efficiency \(\eta_{th}\) and a propulsive efficiency \(\eta_{pr}\)

\[
\eta_o := \eta_{th} \eta_{pr} = \frac{F_n V_0}{P_{th}} = \frac{\text{thrust}}{\text{energy in fuel}}. \quad (1.2)
\]

That is, the efficiency in converting thermal power \(P_{th}\) stored in the fuel into propulsive power \(F_n V_0\) can be broken down into an efficiency \(\eta_{th}\) := \(\dot{E}_{\text{kin}}/P_{th}\) in converting thermal into kinetic power \(\dot{E}_{\text{kin}}\) and an efficiency \(\eta_{pr}\) := \((F_n V_0)/\dot{E}_{\text{kin}}\) in converting kinetic into propulsive power.

Propulsive efficiency \(\eta_{pr}\) can thus be increased by reducing the amount of kinetic power “wasted” in the exhaust. In turbofan engines this is achieved by reducing the exhaust’s net velocity by mixing the core exit flow with the low-momentum bypass stream. The difference in momentum between engine intake and exit is thus achieved by increasing the accelerated mass, rather than excessively raising its exhaust velocity.

\(^3\) Features of lean burn technology have been applied in the TAPS combustor family by General Electric in different in-service engines (Foust et al. [51]).
Thermal efficiency $\eta_{th}$ can be increased either by reducing entropy production $\Delta s$ in the individual components, i.e., increasing component efficiency and approaching the ideal Brayton cycle (Fig. 1.1 b), or by increasing the cycle’s pressure ratio $\pi$. However, it can be shown that an increasing $\pi$ must be accompanied by an increase in turbine inlet temperature $T_{t,40}$ in order to maintain engine thrust (Birch [18]).

1.1.3 Impact on the High Pressure Turbine

The requirement of increasing engine efficiency has led to a steady increase in turbine inlet gas temperatures $T_{t,40}$ as shown in Fig. 1.2 (a). Nowadays, the extreme thermal conditions require active cooling, such as film cooling, and passive cooling, such as thermal barrier coatings (TBC), of the metal components in the HPT (cf. Section 2.3.2).

At the same time, the trend of increasing the bypass ratio $\alpha$ to increase propulsive efficiency $\eta_{pr}$ requires a downsizing of the core engine. Smaller combustor volumes expose the turbine ever more to thermal non-uniformities (hot spots) and residual swirl imposed on the flow in the combustor (Fig. 1.2 b). Therefore, the design of cooling and HPT aerodynamics becomes more dependent on its aero-thermal inlet conditions.

As a consequence of the extreme combustion temperatures, the design of combustor and high pressure turbine nowadays relies heavily on numerical simulations because measurements in the “hot part” of the engine are very difficult. Currently, the only experimental method to assess the thermal load of turbine hardware under real conditions is the application of thermal paint (Clemen et al. [34]).

Hot spots and residual combustor swirl affect turbine aerodynamics in multiple ways. A sophisticated layout of film cooling holes is required to account for the non-uniform inflow. Secondary flows are generated (cf. Sec. 2.4.1) that decrease turbine efficiency $\eta_{HPT}$ (Schmid [157]) which in turn decreases the engine’s overall
Swirl and coherent turbulent structures from the combustor reduce the available flow cross section in the turbine and thus decrease turbine capacity. As shown in Section 2.2.4, this may affect the stability and efficiency of the entire engine cycle.

1.1.4 Analysis of Combustor Turbine Interaction

The downstream effects of combustor exit flow must be accounted for in HPT design. Its influence on aerodynamics and cooling of the first HPT stage is referred to as combustor turbine interaction (CTI). There is also a considerable upstream effect of the stator vanes' potential field on the combustor (Roux et al. [149], Klapdor et al. [96], Vagnoli and Verstraete [176], Cha et al. [31]) which is, however, usually not considered in current engine design processes.

The trends of downsizing the core engine and increasing $T_{t,40}$ elevate the importance of this interaction. Combustor and HPT must be regarded as a physical unit to a greater extent. Currently, the main obstacle in implementing this principle into design processes is a lack of the required numerical tools. The burner and turbine subsystems both show different, highly complex physical phenomena that require specialised modelling approaches.

The flow in the combustor can be regarded incompressible and is highly turbulent due to the generated swirl in lean combustion systems and the cross flow from dilution ports used for quenching in RQL combustors (cf. Fig. 2.1). Moreover, the modelling of the chemical reactions and their complex interaction with flow turbulence requires specialised models.

The flow through the HPT, on the other hand, is mostly compressible. Specialised models are required for the simulation of turbomachinery in order to account for the unsteady interactions between blade rows (Appendix B.2). Also, coupling computational fluid dynamics (CFD) simulations with numerical thermal and stress analyses of the highly loaded rotor blades by means of conjugate heat transfer (CHT) simulations becomes ever more feasible in the design of turbines.

The current industrial approach to modelling CTI is sketched in Fig. 1.3 (left). Both subsystems are analysed separately using different specialised codes. The time-averaged combustor flow field is extracted on an axial plane which is then used as inlet condition to a subsequent HPT simulation. This combustor turbine interface, of which the temperature distribution is referred to as “the traverse”, therefore acts as aero-thermal coupling of the combustor and turbine. Depending on the fidelity of the design approach, the traverse contains a two-dimensional (2D) or a one-dimensional (1D, circumferentially averaged) flow field, or even just a uniform zero-dimensional (0D) field (Povey and Qureshi [135]).

This process is comparably simple and also pragmatic as it allows to maintain the organisational structures of separate combustor and a turbine design teams and
divides the component design into well defined regimes of responsibility. Also, it provides a convenient possibility to scale turbine operating conditions, mainly influenced by $T_{t,40}$, to the design conditions assumed in engine performance calculations. However, it comes with several physical disadvantages. Time dependency is lost at the CTI interface, upstream effects on combustion are neglected and a discontinuity of some flow quantities at the CTI interface occurs. The effect of the latter and its implications are discussed in Section 3.3.2.

![Diagram showing coupled and uncoupled approach to CTI](image)

**Figure 1.3:** Coupled (left) and uncoupled (right) approach to CTI. Aero-thermal conditions from combustor CFD are used as inlet boundary condition for the decoupled turbine simulation.

Different ways to circumvent these shortcomings have been proposed. A promising code coupling approach, that was recently described by several authors, follows the idea of *automatically* coupling the two different, specialised codes in Fig. 1.3 (left) by mutually exchanging information on an overlapping volume between outlet of the combustor domain and turbine inlet. Mass, momentum, energy and turbulent quantities are passed from the combustor CFD domain to nozzle guide vane (NGV) inlet and static pressure is passed back to combustor outlet. The quantities are interpolated to the respective meshes and the two solvers iterate in a *staggered scheme* (Morada [124]), i.e., each solver waits in each time-step until the other one is converged and then updates the respective remote boundary conditions. Salvadori et al. [152] and Insinna et al. [80] report the successful steady code coupling of combustor and first stator row, whereas Vagnoli and Verstraete [176] extended the approach to unsteady coupling. The main challenge is to enable an efficient and accurate interpolation between the two codes.

A second approach is to integrate all the required modelling features from combustor and turbine into a single code as illustrated in Fig. 1.3 (right). This would enable a computation of both components as a single physical system without any coupling involved. The main challenge is to incorporate compressibility into combustion solvers, which are typically established based on low-Mach number assumptions with respect to discretisation, combustion modelling, material properties and boundary conditions. The approach is followed by Klapdor et al. [96] and Raynaud et al.
[142] for the Rolls-Royce combustion solver PRECISE-UNS. Although there are promising advances with both strategies, it is yet unclear when these will reach the required maturity and stability required for an integration into engine development. The focus of this work is on the downstream effects of combustor exit flow and the conducted turbine simulations are of decoupled type.

1.1.5 Uncertainty of Turbine Inflow Conditions

The challenges in modelling the flow through combustor and turbine emphasise the need for measured data for validation, especially at the CTI interface. However, these data are particularly scarce, due to the extreme temperatures, and interface conditions can often only be approximated based on measurements in a vaneless passage downstream of the combustor. The quantitative thermal interface conditions are therefore only known within an uncertainty band which is determined by measurement errors. Montomoli et al. [123] report a typical uncertainty in temperature prediction of ± 2% at 1500 K, even with equipment of highest fidelity.

![Figure 1.4: Turbine inlet traverses measured on a full annular test rig. Data were provided by RRD. The numbers on top give the difference of each sector to all sectors in area averaged mean temperature (ΔT̄) and maximal temperature (ΔT_{t,max}).](image)

Furthermore, the traverse occurring in the engine is not unique due to spatial and temporal\(^4\) variations. Fig. 1.4 shows the time-averaged 2D distribution of turbine inlet temperature of different sectors in a civil engine, tested on a full annular combustor rig such as the one shown by Lazik et al. [110]. Similar results are shown for a military engine by Povey and Qureshi [135]. It can be seen that, due to different installation situations of the burner nozzles, no two sectors are equal but they show a considerable variance in mean and peak temperature. In engine design this variance is typically not accounted for, but only a representative average of all sectors is considered.

Still, even the reliable numerical prediction of average thermal conditions is difficult, as it is largely dependent on the upstream modelling of turbulent mixing. In general,

\(^4\) Temporal variations do not only occur on a micro-time scale due to turbulent fluctuations, but also on larger scales due to thermal expansion of the components and wear of in-service engine hardware which leads to deviations of the component geometry from design conditions.
CFD based on the Reynolds-Averaged Navier-Stokes (RANS) equations and linear eddy viscosity models (EVMs), the current industrial standard, has been shown to inaccurately predict the combustor exit traverse compared to more advanced turbulence models (Boudier et al. [22], Cubeda et al. [38]). The predicted turbine inlet swirl and turbulence distributions suffer equally from insufficiencies in turbulence modelling. Uncertainties associated with swirl prediction are illustrated in Fig. 1.5. The flow field at the CTI interface of the Large Scale Turbine Rig (LSTR, Section 3.4.1) is shown as predicted by measurements and two different flow simulations\(^5\) using scale adaptive (SAS) and the Speziale-Sakar-Gatski (SSG) Reynolds-Stress turbulence modelling (RSM), respectively. It can be seen that the three methods show qualitatively similar results. The quantitative distribution of flow angles yet shows significant differences.

\(\frac{\alpha_{\tan}}{\alpha_{\text{rad}}} (\degree)\)

\(\alpha_{\tan}\) is shown on the left as retrieved from measurements (Exp) and simulations using SAS and RSM turbulence modelling. Circumferential area averages of flow angles are shown on the right.

Measurements of the aerodynamic flow field are less difficult than temperature measurements of the hot flow, but often compromised by the fact that experiments are run on “cold” combustor rigs without chemical reaction which can significantly affect the swirl flow pattern (Section 2.1.2). Even at these cold conditions, probe measurements of swirling flows are challenging because wide calibration maps are required to resolve the large flow angles (Qureshi et al. [140]) and local backflow cannot be resolved.

### 1.2 Placement of this Work, Scientific Approach and Research Objectives

The aforementioned examples illustrate various sources of uncertainty\(^6\) in the predicted CTI traverse. The systematic investigation of causes of uncertainty in traverse prediction is a relatively new field of research. However, the understanding

\(^5\) The SAS was conducted by Hilgert et al. [73], the SSG-simulation was conducted by Bäumler [13].

\(^6\) It is acknowledged that the various effects subsumed to contribute to deviations of the flow conditions yield a somewhat diffuse definition of the term “uncertainty” in the context of this work. However, the scarcity of published data of realistic turbine inlet conditions does not allow for a more thorough classification which remains a task for future research.
of the propagation of uncertainties in the predicted traverse through the HPT is essential because small deviations in metal temperatures have a large impact on turbine life (Section 2.3).

That is, why robust design becomes ever more important by which the uncertain inputs $X$, e.g. combustor clocking, are chosen such that the outputs $Y$, e.g. turbine metal temperatures, have a minimal variance. The aim is to minimise the risk of the technical sub-system to adopt a critical state which would cause a failure of the entire engine. The relation of the variance in $X$ and $Y$ is provided by uncertainty quantification (UQ) methods. The principle of robust design and an approach to UQ in the context of this work are illustrated in Fig. 1.6.

Work in this field has been published by Montomoli et al. [122] in the form of propagation analyses of uncertainties in the thermal field. However, to the author's knowledge no studies investigating the uncertainty propagation of inlet swirl or its interaction with the temperature field under realistic conditions have been published.

![The Principle of Robust Design](image1)

**Figure 1.6:** Robust design approach applied to high pressure turbines with respect to uncertain inflow conditions $X$ (here, the hot spot clocking position $\theta^{\text{hs}}$)

The results of this work are supposed to improve the understanding of aerodynamic phenomena related to CTI from the perspective of a turbine aerodynamics and cooling engineer. The scientific challenge is the isolation and comparable analysis of the different effects that culminate in the highly complex CTI flowfield.

The results of this work are obtained by means of turbine CFD. A flow model is developed to create realistic 2D combustor exit profiles, representative of lean burn combustion systems, based on a set of parameters $X$. These modelled combustor turbine interface conditions are used as inlet conditions to CFD (Fig. 1.6, right). The effects of uncertainties in the parameters $X$ on turbine performance $Y$ are analysed.
by means of statistical UQ methods. Research objectives can be summarised as follows:

- Review of the CFD analysis of turbines and the decoupled simulation of combustor and turbine with regard to uncertainties in the prediction of aero-thermal performance data.
- Quantification of characteristic uncertainties in 2D turbine inlet conditions.
- Development and validation of a flow model yielding realistic, 2D HPT inlet conditions characteristic for lean burn combustors based on a set of model parameters $X$.
- Integration of the model into an efficient uncertainty quantification workflow.
- Quantification of uncertainties in turbine performance $Y$ for the characteristic uncertainties of the parameters $X$ and identification of the most influential parameters in $X$.
- Identification of aerodynamic mechanisms responsible for changes in performance from correlations of the inputs $X$ to the outputs $Y$.

1.3 Thesis Outline

The content of this work is structured as follows. The state of the art is presented in Chapter 2 focussing on jet engine combustors, turbines and their interaction. The methods used are presented in Chapters 3 and 4. In the first part of Chapter 3, an overview on computational fluid dynamics and turbulence modelling is given, and aspects relevant to the simulation of turbomachinery flows and combustor turbine interaction are discussed. The first part concludes with an evaluation of the used flow solver. In the second part, fundamentals of statistics are reviewed and UQ methods are introduced. The developed model for parameterised combustor exit flow is presented in Chapter 4 and evaluated by reproductions of combustor exit flow traverses from different test cases. The integration of this model into an uncertainty quantification workflow is proposed and the underlying assumptions and limitations of the approach are discussed. Applications of this approach are presented in Chapter 5, which is divided in two parts. In the first part, effects of iso-thermal combustor exit swirl on turbine aerodynamics are analysed and correlations of the prescribed turbine inlet conditions with a particular type of induced secondary flows are described. In the second part, the propagation of uncertainty of the inlet conditions through an iso-thermal turbine test rig and a realistic HPT set-up is analysed and the effect of secondary flows on hot streak migration are interpreted making use of the results from part one.
The overall findings of this work are summarised and an outlook on possibilities for future work is given in Chapter 6.
2 State of the Art

2.1 Combustion and Emissions

2.1.1 Geometry and Working Principle of Modern Combustion Chambers

The purpose of a combustor is the transfer of chemical energy, stored in the fuel, to thermal energy in the flow. In order to ensure that an engine can be certified, the combustion is to be efficient with low emissions and noise (Bräunling [24]). In order to grant safety of operation, the flame must be stable and (re-)ignitable. Combustion efficiency is achieved by fast atomisation, evaporation and mixing of the liquid fuel with air, whereas swirling flow hydrodynamically stabilises the combustion (McGuirk [115]). In order to optimise thermodynamic cycle efficiency, pressure loss is to be reduced. Apart from viscous effects, a considerable amount of pressure loss in combustors occurs due to the heat addition (according to the Rayleigh flow model). A secondary requirement is the well-tailored distribution of temperature at the exit to the turbine to minimise thermal stress in the rotor.

The geometry of a typical, modern RQL combustor is schematically shown in Fig. 2.1 (left), consisting of the following elements: A pre-diffusor \( 1 \) reduces the inlet velocity in order to minimise total pressure loss in the combustor and the risk of flame blowout. The volume behind the diffusor is separated by the liner \( 2 \). It encloses the domain of combustion, defining a primary stream for reaching the desired equivalence ratio \( \phi \), Eqn. (2.1), and a secondary stream, entering through a single or multiple rows of ports \( 3 \). The secondary stream cools the liner and is

![Gas path through a modern RQL combustor](image-url)

Figure 2.1: Gas path through a modern RQL combustor (left) and adiabatic flame temperature \( T_{ad} \) over equivalence ratio \( \phi \) for RQL combustors (right, adapted from Blazowski [20]).
mixed with the primary stream in order to quench the mixture to the desired turbine inlet temperature. The fuel injector delivers the liquid fuel stream which is then atomised, vaporised and mixed with air. The combustion of the mixture is initiated by an ignitor. The flow behind the injector is set to a swirling motion by the swirler for flame stabilisation. Part of the air is injected as rear inner discharge nozzle (RIDN) coolant in front of the NGV in order to cool the turbine end walls. The typical position of aero-thermal data transfer between combustor and turbine in design is at the combustor turbine interface.

As mentioned in Section 1.1.2, the formation of pollutants in a combustor depends on its operating condition which is characterised by the equivalence ratio

$$\phi := \frac{\dot{m}_f/\dot{m}_{ox}}{(\dot{m}_f/\dot{m}_{ox})_{stoic}},$$

where $\phi > 1$ corresponds to a rich and $\phi < 1$ to a lean mixture. The equivalence ratio determines the adiabatic flame temperature $T_{f,ad}$. The correlation of $T_{f,ad}$ with $\phi$ is shown in Fig. 2.1 (right). It can be seen that the highest temperatures, which are mainly responsible for the production of thermal NO$_x$, occur at stoichiometric conditions $\phi \approx 1$. Thermal NO$_x$ is lowered in modern low-emission combustions systems by avoiding the high temperatures at $\phi \approx 1$ either by quickly quenching the rich mixture with a large amount of cooler air to a lean state (RQL or “rich burn”) or operating entirely in the lean regime (Lean-premixed-prevaporised, LPP or “lean burn”). Since no quenching is applied in lean burn combustors, they do not require the dilution ports in Fig. 2.1 and the bulk of air (70-80 %) is fed to the combustor directly through the swirler. In general, the volume of lean burn combustion chambers is reduced in comparison to RQL which enhances the aerodynamic coupling of the combustor with compressor and turbine (McGuirk [115]).

Both approaches are capable of reducing NO$_x$ but face different problems (Huang and Yang [76]): Whereas RQL suffers from incomplete mixing and the formation of soot at high $\phi$, the major problem of LPP is the risk of combustion instability and blow out which can lead to system failure.

### 2.1.2 Combustion Instabilities

The stable operation of combustors (lean burn, in particular) is threatened by hydrodynamic and thermo-acoustic instabilities. The latter is not directly relevant in the context of this work and therefore not discussed here.

Traditionally, combustors are designed of diffusion-flame type (fuel and oxidiser are originally separated, as in a candle) due to its reliability and stability (Huang and Yang [76]). Lean combustors, on the other hand, rely on a premixed flame (as in a Bunsen burner). According to a simplified model by Damköhler [41], the speed
at which a flame propagates through a quiescent, premixed gas, is the sum of its laminar flame speed $S_L$ and turbulent fluctuations. Their velocity $u'$ is typically much larger than $S_L$ in gas turbine combustors which emphasises the importance of turbulent mixing for efficient combustion (Farokhi [48]). If the surrounding flow moves with a velocity $u$, that is much different from $S_T$ (above the blowout speed), flame extinction will occur.

In order to prevent flame extinction, the majority of gas turbine combustors uses swirl injectors producing central toroidal recirculation zones (Huang and Yang [76]). The flow velocity $u$ is decreased in the recirculation zone which enables a stable operation and prevents flashback of the flame. The flow starts to recirculate at sufficiently high swirl velocities $u_{sw}$ that cause vortex breakdown. The stable range of a vortex can be assessed by the swirl number\(^1\)

$$S_0 := \frac{\int \rho u_{ax} (u_{sw} R) R dR}{L_{ref} \int \rho u_{ax}^2 R dR} \tag{2.2}$$

which relates the axial fluxes of swirl and axial momentum (Gupta and Lilley [62]). The quantities in Eqn. (2.2) are illustrated in Fig. 2.2 (left). The breakdown, an abrupt reversal of the flow through the vortex core forming a recirculating bubble, occurs at a critical value of $S_0 = 0.6$, due to the forming of two pressure gradients (Huang and Yang [76]): A positive radial gradient balances the centrifugal force of rotation, whereas the flow expansion and decay of swirl cause an adverse axial pressure gradient on the centre line.

\[\text{Swirl Number} \quad \text{Vortex System in the LSTR Swirl Generator} \quad \text{Bubble-Type Vortex Breakdown}\]

\[\begin{align*}
\text{Precessing vortex core (iso-}/\omega\text{)} \\
\text{Recirculation bubble (iso-}u_{ax}\text{)} \\
\text{Residual swirl (iso-}/\lambda_2\text{)} \\
\text{Unstable shear zones} \\
\text{Corner recirculation} \\
\text{Turbine inlet plane}
\end{align*}\]

\[\text{Figure 2.2: Quantities defining the swirl number (left), flow pattern in the LSTR swirl generator (centre, CFD set-up shown in Appendix D.2) and experimentally observed bubble breakdown (right, from Sarpkaya [153])}\]

Sarpkaya [153] observed three different modes of vortex breakdown in confined steady flow, of which the axisymmetric “bubble” breakdown followed by a vortex spiral is closest to the typical flow field in combustion chambers (Fig. 2.2).

\(^1\) Effects of turbulent fluctuations and pressure differences are neglected in this formulation.
A phenomenon, which is special to non-reactive combustor simulators, is the formation of a precessing vortex core (PVC). The PVC is a global, self-excited hydrodynamic instability of the flow, which manifests in a three-dimensional (3D), asymmetrical flow structure precessing around the central zone of flow recirculation in the form of a corkscrew (Huang and Yang [76]), visualised red in Fig. 2.2. It is generally undesired, due to the risk of resonant coupling with acoustic frequencies, but the onset of PVC is still poorly understood (Hall et al. [64]). Oberleithner et al. [130] demonstrated that radial density gradients in real combustion chambers suppress the growth of global unsteady modes that cause the PVC, explaining the findings of other studies (e.g. Roux et al. [148]). That is, the swirl pattern in iso-thermal test rigs may be substantially different from that in reactive combustion chambers due to missing density stratification.

A third component, that completes the typical flow field in a combustor, is the formation of corner recirculation in the shear zones between the main flow exiting the swirler and the ambient fluid (Jacobi et al. [88]).

2.1.3 Combustor Exit Flow Field

According to Lazik et al. [110], the design of industrial combustion chambers is conducted mostly numerically by means of steady RANS calculations, which are accompanied by unsteady RANS and Large Eddy Simulations (LES).

**Figure 2.3:** End wall-induced swirl migration in opposite directions in the MT1 (1 from Insinna et al. [80]) and the LSTR due to different swirl orientations

Validation takes places on test rigs of different technology readiness levels. Povey and Qureshi [135] report a general insufficiency of reliable, open experimental data. From a turbine designer’s point of view, the most important information is the combustor exit flow field (Fig. 1.2 b) which is characterised by unsteadiness,
residual swirl (in case of lean burn combustion), a non-uniform temperature traverse and high turbulence intensity $Tu$.

Turbine inlet swirl is often quantified by the maximal tangential angle $\max(\alpha_{\tan})$ in the CTI traverse that is reported to vary between $30^\circ$ and up to $50^\circ$ in different experimental studies (Giller and Schiffer [54], Qureshi et al. [140], Bacci et al. [5], Jacobi et al. [88], Wilhelm et al. [186]).

A major degree of freedom in designing a HPT for inlet swirl is clocking which describes the circumferential positioning of swirlers relative to the stator vanes. However, depending on the combustor-to-vane count the clocking position may not be constant around the circumference, causing a different 2D inflow condition to each vane.

Moreover, in annular rigs, the swirl clocking position at the turbine inlet is not aligned with the geometrical position of the swirler, but a circumferential movement of the swirl core in the direction of swirl at the casing can be observed (Wilhelm et al. [186], Bacci et al. [7]) as shown in Fig. 2.3. This movement is mainly due to the inclination of the hub end wall between combustor and turbine (cf. Fig. 5.15). Insinna et al. [80] explain the phenomenon on the basis of the steady, inviscid vorticity equation

$$ (u \cdot \nabla)\omega = (\omega \cdot \nabla)u \quad (2.3) $$

that couples gradients in velocity $u$ and vorticity $\omega$. Vorticity is originally aligned with the swirler axis but deflected circumferentially by the axial gradients induced by the hub end wall (Fig. 2.3). In case of parallel end walls, this tangential movement does not occur (cf. Giller and Schiffer [54], Jacobi et al. [88]).

Aerodynamic measurements by means of a passive scalar tracing technique at the exit of a real-engine RQL combustor (without reaction, at reduced Mach and Reynolds number) under the influence of NGVs are reported by Cha et al. [30]. They found, that large turbulent length scales are at the order of the spacing of the dilution ports rather than the diameters of the ports or injectors. Cha et al. [31] report $\overline{Tu} = 35\%$ behind the combustor. Turbulence levels behind combustor simulators are reported by Bacci et al. [6] ($\max(Tu) = 28\%$ and $\overline{Tu} = 17\%$), Wilhelm et al. [186] ($\max(Tu) = 45\%$ and $\overline{Tu} = 25\%$) and Beard et al. [15] ($\overline{Tu} = 11\%$). Note, that measurements of turbulence by means of hot-wire probes, can only be conducted under iso-thermal conditions. Optical measurements behind a reactive lean burn combustor at two realistic operating points were conducted by Schroll et al. [162]. Peak turbulence values were at $15\%$ and $20\%$, respectively.

Circumferential variations of temperature behind RQL combustors are caused by the discrete injection locations of fuel and dilution air (3 and 5 in Fig. 2.1) and radial
variations behind lean burn combustors by liner coolant. *Effusion cooling*\(^2\) is used for cooling of liners in novel LPP combustors, which leads to highly different turbulent length scales between the main swirling flow and the near wall flow (McGuirk [115]).

According to Goebel et al. [55], raising the swirl number \(S\) increases radial non-uniformity, whereas an increasing dilution flow rate flattens the radial profile. In general, a flatter radial temperature profile can be observed for lean burn combustors (Werschnik et al. [182]). 2D temperature exit profiles are generally hard to compare between combustors and test rigs at different operating conditions. The non-uniformities are therefore quantified by (local) overall (OTDF) and radial (RTDF) temperature distortion factors\(^3\)

\[
\text{(L)OTDF} := \frac{T_{t,40(\text{max})} - T_{t,40}}{T_{t,40} - T_{t,3}} \quad \text{(L)RTDF} := \frac{T_{t,40(\text{circ})} - T_{t,40}}{T_{t,40} - T_{t,3}}.
\]

The OTDF quantifies *absolute* non-uniformities, that primarily affect the stator heat load, whereas *circumferentially mixed-out* non-uniformities quantified by the RTDF are relevant for the rotor. TDFs apply approximately equally to static and total temperatures and are almost independent on engine operating conditions (Cha et al. [31]). A common alternative to these distortion factors is the temperature ratio \(T_{t,40}/T_{t,40}\), which is a good indicator of most relevant phenomena and proposed as universal indicator by Povey and Qureshi [135]. Kouper et al. [99] report this quantity to range between 0.75 and 1.15 for several hot streak simulator facilities. Behind the first nozzle, the non-uniformities in temperature as well as the high swirl angles and turbulence intensity are reduced by the acceleration of the flow through the vanes (Farokhi [48]).

### 2.2 Turbine Aerodynamics

#### 2.2.1 Work Extraction in Turbines

The turbine is located downstream of the combustor. Its purpose is to extract work from the flow which is used to drive the compressor mounted to the same shaft (cf. Fig. 1.1 a). In most jet engines turbines are arranged axially, to enable high mass flows, with multiple stages (stator-rotor pairs). Turbines in modern civil aircraft engines consist of two or three sections, referred to as *high*, *intermediate* and *low pressure turbine*, respectively. These sections are mechanically decoupled since they

\(^2\) The material is perforated by many micro-scale holes in order to enable a homogeneous film.

\(^3\) The superscript “circ” denotes a circumferential average of the temperature profile. The subscript “max” indicates the maximum value of the respective quantity. For the local variants (L) of the TDFs, the subscript “max” does not apply. The quantities are referred to as pattern and profile factor in the US, Povey and Qureshi [135].
run on different spools with different speeds of rotation $\omega$, each connected to the corresponding section of the compressor (cf. Fig. 1.1 a). This allows for lower rotational speeds of the fan to prevent noise and shock loss. High pressure turbines are operated transonically and consist of one or two stages. The number of stages is a trade-off between the weight of the turbine and the efficiency deficit from high Mach numbers (Bräunling [25]).

The change of energy in the fluid results from the transfer of momentum in circumferential direction to the rotor. Stators are merely used to turn the flow direction, thus providing an inlet swirl component for the rotor. As can be seen in the overall engine cycle in Fig. 1.1 b), the pressure drops between turbine inlet and outlet. Turbine blades are therefore designed with decreasing cross-sectional areas in the $x$-$\theta$-plane in order to accelerate the flow\(^4\). Modern rotor blades employ reaction blading with cross-sections of decreasing area. The rotor is thus driven not only by the circumferential turning of the flow but, to a large part, by its acceleration in the passage.

![Blades and Velocity Triangles](image)

**Figure 2.4:** Velocity triangles (left), thermodynamic states and extracted power (right) in a turbine stage

The extracted specific work is proportional to the speed of rotation $U = \omega r$ and the deflection in the rotor $\Delta C_u$ (cf. Fig. 2.4) according to the Euler turbine equation

$$\frac{P}{m} = \Delta h_t = U_2 C_{u,2} - U_1 C_{u,1} \approx U \Delta C_u,$$

(2.5)

where $U \Delta C_u$ is evaluated at midspan. Eqn. 2.5 can be derived from the axiomatic assumption of conservation of angular momentum and assuming a steady state, a homogeneous distribution of swirl along the radius, $r C_u =$ const., and a flow that is ideally guided by infinitely small rotor blade spacing (there is no deviation between flow and metal angle).

\(^4\) Turbine stators are therefore commonly referred to as nozzles.

2.2 Turbine Aerodynamics
2.2.2 Efficiency and Losses

Turbine Efficiency

The *isentropic efficiency* $\eta$ of a turbine is defined as the ratio of power $P$ extracted from the flow through the turbine to the power $P_i$ of an ideal, isentropic reference process. The powers are illustrated in Fig. 2.4 (right). Assuming an adiabatic system, these are equal to the respective changes in total enthalpy $\Delta H_t$ according to the first law of thermodynamics. Assuming a constant heat capacity $c_p$, the isentropic enthalpy difference can be replaced by temperature differences. The unknown isentropic outlet temperature can then be expressed by pressures (using the isentropic relation) yielding an expression for $\eta$ which depends only on fluid properties and quantities that can be measured in the flow

$$
\eta := \frac{P}{P_i} = \frac{\Delta T_t}{T_{t,in} \left[ 1 - \left( \frac{p_{t,out}}{p_{t,in}} \right)^{\frac{\gamma-1}{\gamma}} \right]},
$$

where $\Delta T_t = T_{t,in} - T_{t,out}$. The definition of an isentropic efficiency is unambiguous in case of an un-cooled turbine stage. If turbine cooling is considered, however, there are several possible definitions of the ideal power $P_i$ as explained by Young and Horlock [189]. The relevant question is, if and how mixing of coolant with the main flow is to be considered a loss. A common definition is the one by Hartsel [67]

$$
\eta := \frac{\sum_j \dot{m}_{in,j} \bar{c}_{p,j} \Delta T_{t,j}}{\sum_j \left\{ \dot{m}_{in,j} \bar{c}_{p,j} T_{t,in,j} \left[ 1 - \left( \frac{p_{t,out}}{p_{t,in,j}} \right)^{\frac{\gamma_{j}-1}{\gamma_{j}}} \right] \right\}}
$$

assuming that no mixing occurs in the ideal process, but all streams $j$ (coolant and main flow) separately expand isentropically to a common exit pressure $p_{t,out}$. Under realistic conditions, the fluid properties $c_p$ and $\gamma$ vary between the main stream and coolant due to their different temperature levels. Therefore, individual values of $\bar{c}_{p,j}$ and $\bar{\gamma}_{j}$ are used for each stream (averaged between each inlet $j$ and outlet). It can be shown, that this leads to “real gas” problems of the Hartsel efficiency: It varies – just by definition – with the fraction of coolant mass, if fluid properties are not constant (Young and Horlock [189]). Alternative definitions were therefore proposed, assuming an adiabatic mixing of all streams, which are then expanding to $p_{t,out}$ as a single, mixed-out stream. The *mixed-out efficiencies* in the literature differ in the definition of pressure of the mixture. In this work, the Hartsel efficiency according to Eqn. (2.7) is used because no variations of cooling mass flow are conducted and changes in mainstream mass flow are assumed negligible. The implementation of efficiency computation in the used flow solver is described in
Appendix A.1.2.

It can be shown that, due to a decrease of inlet temperature to each turbine stage, the averaged isentropic efficiency of a stage is lower than the isentropic efficiency of the multi-stage turbine (Bräunling [25]) which is the reason for the usage of a polytropic efficiency \( \eta' := \delta h_t / \delta h_{t,i} \). The two efficiencies are related by

\[
\eta = \frac{1 - \tau}{1 - \tau^{1/\eta'}},
\]

(2.8)

where \( \tau := T_{t,\text{out}} / T_{t,\text{in}} \). When interpreting efficiency changes \( \Delta \eta \) in the context of this work, it is important to note that these are not synonymous with changes in entropy loss \( \Delta s \). As shown in Appendix A.1.1, variations of 2D inlet conditions inevitably cause changes in power \( P \) that affect \( \Delta \eta \), even if \( \Delta s = \text{const.} \).

According to Denton [43], a metric, suitable to quantify the entropy increase across a turbine stage, is the entropy loss coefficient \( \zeta_s \). It is often desirable to separately account for losses in the stator, where no work is done, \( P = 0 \). If the flow is considered adiabatic, total enthalpy is conserved, \( H_t = \text{const.} \), and only total pressure losses need to be accounted for. They are usually normalised by the dynamic head in a pressure loss coefficient \( \zeta \). The two loss coefficients are defined by

\[
\zeta_s := \frac{T_{\text{out}}(s_{\text{out}} - s_{\text{in}})}{h_{t,\text{out}} - h_{\text{out}}} \quad \text{and} \quad \zeta := \frac{p_{t,\text{in}} - p_{t,\text{out}}}{\frac{1}{2} \rho_{\text{in}} u_{\text{in}}^2},
\]

(2.9)

respectively. All quantities in Eqn. (2.9) are mass flow-averaged on the respective interfaces. In this work, additionally, the local equivalent \( \zeta(r) \) is analysed, where outlet pressure \( p_{t,\text{out}} \) is mass flow-averaged in circumferential direction only.

Losses in Turbines

Historically, losses in turbomachines are broken down into three groups, although these cannot be regarded fully independent (Denton [43]). Profile losses describe losses at the 2D blade profiles, i.e., friction in the boundary layers, mixing at the trailing edge and losses due to flow incidence

\[
\Delta \alpha_{\text{tan}} = \alpha_{\text{tan}} - \alpha_{\text{tan,DP}},
\]

(2.10)

a misalignment of the flow at the leading edge (LE, cf. Fig. 2.4) as in classical airfoil aerodynamics. Secondary flow loss covers 3D flow effects introduced by the presence of the hub and shroud end walls. The loss from flow over the blade tip is referred to as leakage loss. Each of the three groups contributes roughly 1/3 to the overall losses in HPTs (Denton [43]).

Additional sources of losses in turbines are shocks at the trailing edge of transonic turbine vanes, entropy generation from heat transfer to solid walls and thermal...
2.2.3 Turbine Secondary Flows

Secondary flow losses describe the transfer of kinetic energy into secondary kinetic energy of flow components perpendicular to the primary flow direction, a part of which is dissipated. This transfer is driven by different mechanisms. According to the classical model by Hawthorne [68], the deflection of inlet vorticity, which is perpendicular to the main flow (as in boundary layers), induces a secondary circulation (Fig. 2.5 a). Although the vorticity is commonly caused by friction in the wall boundary layers, the effect itself is inviscid and can be explained by a momentum balance of boundary layer flow. The transverse pressure gradient \( \frac{dp}{dy} \) from suction side (SS) to pressure side (PS) balances the local centrifugal force on a fluid particle with a local radius of curvature \( R_{\text{curv}} \), hence

\[
\frac{dp}{dy} = \rho \frac{u^2}{R_{\text{curv}}} \Rightarrow R_{\text{curv}} \propto u^2.
\]  

(2.11)

As the pressure gradient through the boundary layer is approximately zero, the left hand side of Eqn. (2.11) is constant in the direction of the blade span close to the end walls. Hence, the local radius of curvature \( R_{\text{curv}} \) must increase with wall distance, which imposes a large-scale circumferential motion on the main flow (blue in Fig. 2.5 a).

This inviscid structure interacts with other vortices, that emerge in corners of the geometry due to viscous effects (Fig. 2.5 b). The horseshoe or leading edge vortex originates from flow separation at a saddle point in the conjunction of vane leading edge and end walls (blue). It rolls around the vane and is split into a PS and SS leg.

**Figure 2.5:** Turbine vane secondary flows
The PS leg is energised by the large scale circulation in the passage, described above, forming a structure referred to as passage vortex (red). The merging of boundary layers from vane and end wall surfaces leads to the formation of corner vortices. The different phenomena have been investigated experimentally and multiple approaches to modelling the interaction of these vortices have been proposed. Comprehensive reviews are given by Sieverding [167] and Langston [104]. It is acknowledged in literature that secondary flows in annular sectors are different from linear cascades because, in general, less cross-passage flow is generated (Qureshi et al. [140]). Also, it is reported that the classical picture of secondary flows in Fig. 2.5 is altered by engine realistic operating conditions (Thomas et al. [174]) and the effects of inlet swirl (Pylouras et al. [137]).

The inviscid flow patterns in Fig. 2.5 a) form as long as a vorticity component transverse to the main flow is present. Additional vorticity is generated in curved channels by a radial gradient in total pressure at the inlet, as shown by Hawthorne [69] for inviscid, compressible flow.

A variety of ways to reduce secondary flow losses has been proposed, such as adjusting the LE geometry (Sauer et al. [154], Becz et al. [16]), changing lean and bow of the blade or contouring the passage end walls (Germain et al. [52], D'Ippolito et al. [45]). Wang et al. [179] even propose an entirely novel design concept for turbine vanes that eliminates all corners in the flow path and therefore suppresses the formation of most secondary flows in Fig. 2.5 b).

### 2.2.4 Turbine Capacity

The mass flow through the orifice of a tank under pressure is limited by a critical pressure ratio $\pi_{\text{crit}}$ to the surrounding. In a similar manner, the mass flow through a jet engine’s core is limited by the smallest cross section in the hot part of the gas path. Typically, this throat section is located in the HPT stator. If $\pi_{\text{crit}}$ is exceeded, the flow Mach number reaches unity in this cross section and the turbine is choked.

The maximum corrected core mass flow, the capacity of the turbine

$$\Gamma := \frac{\dot{m}_{40} \sqrt{T_{t,40}}}{P_{t,40}}, \quad \text{(2.12)}$$

is then invariant to a reduction of exit pressure. The effective cross section limiting $\Gamma$ is not only determined by the geometry of the turbine but also by aerodynamic blockage due to secondary flows, turbulent structures and residual combustor swirl. The determination of $\Gamma$ is a crucial element of aerodynamic HPT design because it affects the throttling and pressure rise of the compressor $\pi$ which is an important parameter for the thermal efficiency of the global engine cycle (Section 1.1.2). From continuity of mass flows through compressor and turbine it can be shown that $\pi \propto \sqrt{T_{t,40}/\Gamma}$ (Farokhi [48]).
2.3 Turbine Heat Transfer and Cooling

2.3.1 Heat Transfer

Turbine blades are manufactured of nickel based super-alloys which are subject to extreme thermal conditions. The metal temperature is the most important parameter to assess the life of a HPT which is limited by thermal stresses or fatigue, sulfidation and creep. All of these processes depend either on temperature level or homogeneity. In particular, the creep process rate $R_{cr}$ grows exponentially with temperature (Larson [105])

$$R_{cr} = C \exp \left( -\frac{\Delta H}{RT} \right),$$  \hspace{1cm} (2.13)

where $\Delta H$ is the creep activation energy. At typical temperature levels in HPTs, creep life time is halved by an increase of approximately 10 K (Bräunling [25]). A method to translate the uncertainty in metal temperature into an uncertainty in turbine life time can be found in Montomoli et al. [123]. Also, higher temperatures generally cause an increase in maintenance costs due to the advanced materials involved (Bräunling [25]).

The metal temperature results from heat which is transferred from the hot external flow to the metal by forced convection (which is to be diminished in design), conducted within the metal and further convectively transferred to secondary air flows through the blades (which is to be enhanced) that are finally ejected back into the main flow as cooling film. In this process, the metal temperature is not only dependent on the local temperatures of the air streams but also on the resistance against heat transfer between fluid and solid which mainly depends on the local turbulence intensity, thickness of thermal and velocity boundary layer and wall roughness (Bons [21]). It is quantified by the heat transfer coefficient (HTC) $h$ of heat flux $\dot{q}$ from external flow to the wall, as defined by Newton’s law

$$\dot{q} = h(T_w - T_{ref}).$$ \hspace{1cm} (2.14)

On the surface of a HPT vane, highest external HTC can be observed at the LE, where incoming flow impinges on the vane, and towards the TE, where flow acceleration shrinks the flow boundary layer (cf. Fig. 4.11). HTC on the end walls typically increases towards the throat where the flow is accelerated (Qureshi et al. [140]). The reference temperature $T_{ref}$ in Eqn. (2.14) is the far field or bulk temperature of the main flow $T_\infty$.

As pointed out by Moffat [121] the local adiabatic wall temperature $^{5}T_{w,ad}$ is a more appropriate reference for complex internal flows yielding an adiabatic HTC $h_{ad}$. Film

---

5 This temperature is above $T_\infty$ due to viscous dissipation in the boundary layer. The temperature increase towards the wall drives heat away from the wall and the ratio of the resulting tempera-
cooling measurements by Metzger et al. [120] indicate a linear relation of \( h \) with the non-dimensional temperature \( \theta \) which was generalised by Eckert [47] to

\[
h(\theta) = h_{ad}(1 - \eta_{FC} \theta)
\]

and the film cooling effectiveness \( \eta_{FC} \), Eqn. (2.17). The linearity in this relation enables a convenient way to determine the adiabatic HTC from two CFD simulations (or experiments) at two arbitrary temperatures \( \theta \) (Fig. 2.6). Usually, in turbine CFD adiabatic walls and a fixed wall temperature \( T_w \) are set as boundary condition (BC) which yield the distributions of \( \dot{q} \) and \( T_{ref} = T_{w,ad} \), in Eqn. (2.14), and thus \( h_{ad} \). An assumption inherent to Eqn. (2.15) is that HTC in the turbine passage is determined by aerodynamics only. That is, \( h_{ad} \) is assumed to be independent of \( \theta \) and thus of the wall temperature \( T_w \) (Gritsch et al. [59]). Recent studies by Maffulli and He [113] have shown that the influence of \( T_w \) on the local Reynolds number in the boundary layer is not always negligible and \( h_{ad} \) can thus be significantly affected by \( \theta \). The common industrial approach to resolve effects of \( T_w \) is an iterative one-way thermal coupling in which CFD of the fluid and finite element analysis (FEA) of the solid are run iteratively in order to update the wall temperature \( T_w \). A promising alternative is the use of CHT, which integrates CFD and FEA, enabling a two-way coupling in a single simulation. However, large disparities in time scales between convection in the fluid and heat conduction in the solid confine this method mostly to steady analyses.

Since the actual value of \( h_{ad} \) is not of primary interest in the present studies, but rather its change with different HPT inlet conditions, thermal conduction in the metal is not considered. The previously described iterative process is not conducted, and the mean wall temperature \( T_w \) in Eqn. (2.14) is guessed from the thermal condition of the main flow. The resulting HTC distribution is reported in non-dimensional form as Nusselt number

\[
\text{Nu} := \frac{hL_{ref}}{k} = \frac{\dot{q}L_{ref}}{k(T_w - T_{w,ad})},
\]

where \( L_{ref} \) is a characteristic length (the NGV’s axial chord length \( c_{ax} \)) and \( k \) is the fluids thermal conductivity.

---

The temperature difference to the ideal temperature increase from an adiabatic deceleration of the main flow is given by the recovery factor (Schlichting and Gersten [156])

\[
r := \frac{T_{w,ad} - T_{\infty}}{T_{T,\infty} - T_{\infty}} = f(\text{Pr}, \text{Re}).
\]

2.3 Turbine Heat Transfer and Cooling
2.3.2 Film Cooling

As outlined in Section 1.1.3, an increase of turbine inlet temperature $T_{t,40}$ is beneficial for the cycle process and nowadays $T_{t,40}$ is well above the melting temperature of the alloys used (cf. Fig. 1.2 a). In order to reduce heat loads to the components, different technologies have been introduced to enhance the limited capabilities of internal convective cooling: Coolant is bled through the blade surface to form a protective film between hot gas and metal which is additionally protected by a layer of TBC that allows for a raise in external surface temperature and thus reduces the heat flux to it (Bunker [26]). Since the LE region of HPT vanes is strongly curved, it is especially hard to cool with an attached cooling film. That is, why this region is usually heavily cooled by many holes in a showerhead arrangement. An extensive review of research with respect to turbine cooling can be found in Han et al. [65].

In almost all current applications, vane film cooling is supplied through discrete holes, rather than continuous slots, for structural reasons. The layout of the holes is designed to ideally provide a coherent film (as would result from a continuous injection). Improvements have been made by the application of fan-shaped holes with an expanded exit cross-section that decrease the velocity of the ejected coolant. The fundamental aim of film cooling is to achieve the highest possible overall adiabatic film cooling effectiveness

$$\eta_{FC} := \frac{T_\infty - T_{w,ad}}{T_\infty - T_c}$$

with the lowest possible penalty in the thermodynamic cycle (Bunker [26]). According to Denton [43], an increase in cooling mass flow by 1 % causes the overall cycle efficiency to decrease by 1 % which illustrates the necessity to reduce the amount of used coolant to a minimum. Also, the manufacturability of the cooled blades with sufficient minimal wall thicknesses must be granted within tight tolerances. Bunker [27] analysed the effect of geometrical properties of cooling holes on component life and found that a variation of 10 % in cooling hole diameter can reduce the blade life by 33 %.

Performance of individual cooling holes is influenced by their geometrical properties (diameter, ejection angle, hole spacing), wall roughness, external pressure gradients, the local turbulence intensity $T_u$, interaction with secondary flows and unsteady wakes as well as operating conditions. The latter are quantified by velocity and density ratio (DR) of coolant to main flow, the mass flow ratio (MFR), blowing rate (BR),

$$\text{DR} := \frac{\rho_c}{\rho_\infty}, \quad \text{MFR} := \frac{\dot{m}_c}{\dot{m}_\infty}, \quad \text{BR} := \frac{\rho_c u_c}{\rho_\infty u_\infty}$$

(2.18)
and momentum ratio. The effects of the different parameters have been investigated in flat plate experiments. A comprehensive overview is given by Baldauf et al. [9, 10]. In general, high DR and BR improve film cooling protection, unless the cooling film lifts off the surface (Han et al. [65]). It is well known, that a characteristic structure of two counter-rotating kidney vortices forms as a consequence of the interaction with coolant and main flow which lifts coolant away from surface and sucks hot flow under the film layer. The intensity of the structure increases with BR. Novel film cooling hole designs aim at suppressing the formation of kidney vortices (Dhungel et al. [44]). Furthermore, the flow within angled cooling holes shows considerable separation leading to the development of a jet-wake structure within the holes (Leylek and Zerkle [112]), altering their BR. When evaluating a specific layout of cooling holes, the consideration of $\eta_{FC}$ alone is not sufficient. As pointed out by Baldauf et al. [9], configurations with high $\eta_{FC}$ in general coincide with those yielding a high external HTC. A meaningful assessment can therefore only be achieved when regarding the combined effect of $\eta_{FC}$ and $h_{ad}$ on the actual metal wall temperature $T_w$.

Also, the interaction of cooling holes with pressure gradients induced by the vanes must be considered in design. Thomas et al. [174] and Werschnik [180] showed, that the potential effect of NGVs affects the performance of RIDN cooling holes at different circumferential positions.

### 2.4 Combustor Turbine Interaction

#### 2.4.1 Temperature Induced Secondary Flows

Substantial theoretical understanding has been achieved on the effect of hot streaks on secondary flow generation in the turbine. The three major components are summarised in this section.

The substitution principle describes a similarity solution of compressible flows published in 1947 that recently found application in the context of CTI. It was shown by Munk and Prim [126] that in a steady and isentropic flow of ideal gas, changes in the inlet total temperature field of a stator row do not alter streamline pattern, Mach number and total pressure in the stator, as long as the inlet total pressure field (and the bounding geometry) is unchanged. That is, changes of the inlet stagnation temperature distribution to a turbine stator do not create or change secondary flows

---

6 A change of inlet temperature does change the local velocity distribution but in such proportion that the local Mach numbers remain unchanged. An intuitive explanation is given by Barankiewicz et al. [11] who expressed the conservation equations of mass and momentum in terms of Mach number and pressure. The resulting equations are decoupled from the energy equation.
in the stator. There is, however, a significant influence of inlet temperature distortions on secondary flows in the rotor (Povey and Qureshi [135]). In the literature, the effects are explained based on a version of the steady transport equation of vorticity $\omega_s$ in streamwise direction $s$, derived by Hawthorne [70]. An extension can be found in Lakshminarayana [103]. Butler et al. [28] produced a simplified equation by neglecting the small axial derivatives in turbines and assuming an axisymmetric flow to the rotor,

$$R_{\text{curv}} \frac{\partial}{\partial s} \left( \frac{\omega_s}{W} \right) = \frac{2}{\rho W^2} \left( \frac{\partial p_{t,\text{rel}}^*}{\partial r} + \frac{U^2}{2} \frac{\partial \rho}{\partial r} \right), \quad (2.19)$$

where $p_{t,\text{rel}}^* = p_{t,\text{rel}} - 1/2 \rho U^2$. The equation shows that vorticity (hence, secondary flow) is generated in a turbine rotor by radial gradients of both, the rotary total pressure $p_{t,\text{rel}}^*$ and density $\rho$ at stator exit. Both of these gradients are influenced by the combustor exit temperature profile. Typical secondary flow structures, induced by these gradients, are shown in Ong and Miller [133].

Hot streaks emanating from a turbine stator impinge preferably on the PS of a downstream rotor in a positive jet. The phenomenon is referred to as segregation effect, preferential heating or Kerrebrock-Mikolajczak effect (Butler et al. [28]). Kerrebrock and Mikolajczak [94] developed a transport theory describing the influence of temperature wakes from compressor stators on the flow in a rotor row downstream. The effect can be explained, quite simply, by regarding the velocity triangles at different temperatures as shown schematically in Fig. 2.7. Velocity magnitude in the absolute frame increases with temperature which turns the flow vector in the relative frame, moving with rotational speed $U$, towards the PS.

---

7 The effect is confirmed by Butler et al. [28], Salvadori et al. [151], Beard et al. [14] and Khanal et al. [95]

8 The effect is confirmed by Butler et al. [28], Shang and Epstein [165], Salvadori et al. [151], Qureshi et al. [139], Ong and Miller [133]. Gundy-Burlet and Dorney [61] and He et al. [72] found that preferential heating is dependent on the clocking position of the hot spot and may be attenuated by aligning the hot spot with the NGV’s LE. Khanal et al. [95] point out that the attenuation does not occur, if a hot spot and swirl are simultaneously imposed as inlet condition.
2.4.2 Experimental and Numerical Studies

Hot Streaks

Experimental Facilities: Test rigs for the investigation of hot streaks have been built since the 1980s, mostly in the US (Koupper et al. [99]). Almost all rigs in service are non-reactive. Hot streaks are simulated by either locally heating the flow with heat exchangers or mixing streams of gas of different temperatures. The facilities are reviewed by Povey and Qureshi [135]. However, a comprehensive comparison of experimental results is difficult since many rigs struggle to generate a well defined, reproducible temperature profile at turbine inlet.

Investigations at the Oxford Turbine Research Facility: A comparatively novel rig is the Oxford Turbine Research Facility (OTRF) – formerly QINETIQ Isentropic Light Piston Facility (ILPF) and Turbine Test Facility (TTF) – a short duration blow-down rig, equipped with a temperature distortion generator. The most frequently used turbine geometry for CTI studies is the MT1 unshrouded, high-pressure research stage which was developed by ROLLS-ROYCE and tested in the ILPF within the European project TATEF (Chana and Singh [32]) in different cooled and un-cooled configurations.

He and Haller [71] and He et al. [72] examined the effect of hot streak circumferential length scale on the MT1 rotor. Clocking between hot spots and NGVs showed to be most influential, if their count is equal. If there are fewer hot spots than NGVs, the clocking effect is attenuated but the maximal wall temperatures and unsteady forcing of the rotor are increased.

Qureshi et al. [139] evaluated changes in the MT1 rotor heat transfer due to an inlet hot streak. HTC at the casing was almost unaffected (changes below 8 %) but $T_{w,ad}$ at the casing was significantly lowered, caused by a reduction of inlet temperature at the casing to maintain mean inlet temperature $T_{i,in}$. Heat flux on the vane PS was increased, driven by changing $T_{w,ad}$ due to preferential heating rather than changes in HTC.

In a similar study, Beard et al. [14] quantified efficiency reduction and capacity changes due to a hot streak. Additional losses were explained by rotor off-design incidence. The hot streak greatly changed the radial distribution of work extraction in the rotor. The largest differences were found at the casing.

Salvadori et al. [151] numerically reproduced MT1 efficiency measurements (at uniform inlet conditions) and vane loading with and without inlet distortions. A larger effect on HTC was observed on the vane SS than on the PS. Lower HTC was observed at the rotor tip due to lower casing temperatures at turbine inlet, in accordance with Qureshi et al. [139].

2.4 Combustor Turbine Interaction
Other Investigations: Early experiments on the diffusion of hot streaks through vane passages were conducted by means of CO₂ seeding as reported by Butler et al. [28] who essentially confirmed the theoretical predictions of Section 2.4.1. Shang and Epstein [165] investigated the impact of temperature distortion on rotor heat load. They found a migration of hot fluid towards the rotor hub. Similar studies were conducted by Gundy-Burlet and Dorney [61]. Ong and Miller [133] numerically investigated the unsteady migration of hot streaks and formation of secondary flows in the rotor of a low speed research turbine. Tallman [170] investigated costs and benefits of unsteady multi-row simulations from an industrial perspective. He found a strongly increased mixing of a hot streak behind an HPT exit strut (thus better agreement with experiments) in unsteady computations as compared to steady ones. Basol et al. [12] numerically investigated the radial migration of hot gas due to hot streak induced secondary flows and the radial velocity field in the stator. The hot streak clocking position was found to influence the radial distribution of rotor heat load. Clocking the hot streak to the stator pressure side was found to reduce the rotor blade tip temperature. Jenkins et al. [89] report a significant effect of the main flow turbulence level on the decay of a hot streak in a vane passage. At high turbulence (Tu = 20 %), the peak temperature was reduced by 20 %. The addition of film cooling further reduced the hot spot strength resulting in an large attenuation of 75 %. Jenkins and Bogard [90] report a dependence of the hot spot attenuation on its clocking position. It is split in two streaks on the PS and SS which are attenuated differently.

Inlet Swirl and Turbulence

Impact of Inlet Swirl and Turbulence: While hot streaks were shown to primarily affect the flow through the rotor, the impact of residual swirl is mostly confined to the stator due to flow acceleration in the vane passage (Beard et al. [15], Schmid [157]). There is a consistent agreement in the literature on some effects of inlet swirl on stator aerodynamics (Qureshi et al. [140]): Incidence induced by inlet swirl causes a shift of the stagnation line with respect to uniform inflow. The shift is greatest close to the end walls and its direction depends on swirl rotation direction. Positive incidence increases aerodynamic vane loading and leads to higher losses whereas negative incidence reduces them⁹. Moreover, residual swirl that migrates through the stator was found to induce upwash and downwash on the adjacent vanes, according to its sense of rotation, which influences film cooling, the convection of hot streaks and local HTC. Furthermore, an augmentation of vane surface heat transfer with increasing turbulence intensity at combustor exit is reported by

---
⁹ The effect is reported by Giller and Schiffer [54], Schmid et al. [159], Beard et al. [15], Hilgert et al. [73], Jacobi et al. [88]
different authors, such as Ames et al. [1], Radomsky and Thole [141] and Nasir et al. [128].

Swirl Induced Secondary Flows: Jacobi et al. [88] investigated the formation of unsteady flow phenomena, induced by inlet swirl, in a linear, subsonic vane cascade experimentally and by means of LES. They observed the formation of two counter-rotating structures at the vane LE that form due to a “rolling-up” of high momentum fluid into the low pressure core at midspan, as shown in Fig. 2.8. The two structures are convected to the vane’s SS and PS, similar to the horseshoe vortex. In the passage, they are supported or weakened by inlet swirl, depending on its orientation. The authors also state that the shift of the LE stagnation line causes an acceleration and deceleration of flow around the LE towards PS and SS, respectively, and thus radial gradients of static pressure in the passage emerge (shown on the right in Fig. 2.8). These convect the induced structures towards the hub or tip, depending on the swirl orientation. The phenomenon was found to be highly unsteady and the structures fluctuated in radial direction. Furthermore, transient fluctuations of inlet swirl angle by ±15° were identified, which led to fluctuations of the circumferential positions of the hub horseshoe vortex.

Figure 2.8: Swirl induced vortex structures at stator LE (adapted from Jacobi et al. [88])

The radial migration of inlet swirl in the passage, described by Jacobi et al. [88], is similarly observed in the simulation of an industrial dry low-emission combustor by Turrell et al. [175].

Investigations at the Large Scale Turbine Rig: The LSTR at TU Darmstadt is an iso-thermal test rig devoted to study the influence of turbine inlet swirl on aerodynamics and cooling. A detailed description of the rig can be found in Section 3.4.1. The swirling inflow to the turbine is shown in Fig. 2.2. The flow through the test turbine has been examined in different studies.

Werschnik et al. [183] described the flow field in the NGV row based on experimental and numerical data which is summarised in Section 5.3.1. Werschnik et al. [181] investigated the effect of inlet swirl on film cooling effectiveness $\eta_{FC}$ and Nusselt
number $\mathrm{Nu}$ at the end wall of the stator. Film cooling effectiveness decreased significantly by up to 35% with swirl, whereas the increase in $\mathrm{Nu}$ was lower (10-20%). Turbulence intensity was found to increase to $\overline{\mathrm{Tu}} = 30\%$ (max$(\mathrm{Tu}) = 45\%$) with swirl, compared to 1% with axial inflow. Also, NGV pressure loss increased and the flow at stator exit was more mixed-out by swirl. No upstream effect of the NGV on turbulence distribution was recognised. Inlet swirl did not influence traces of coolant flow at sufficiently high MFR (the effect of MFR variation dominated that of inlet swirl).

Werschnik et al. [182] investigated the robustness of the hub cooling scheme with respect to swirl. It was found that swirling inflow changes the location of transition from low to high heat transfer in the passage. The differences in $\mathrm{Nu}$ between adjacent passages with inlet swirl were small, those of $\eta_{\mathrm{FC}}$ were larger. It was concluded that $\mathrm{Nu}$ distribution is relatively robust with respect to swirl (at sufficiently high MFR) whereas $\eta_{\mathrm{FC}}$ is less robust and should be the target when optimising a turbine for inlet swirl.

Schmid et al. [159] numerically investigated the effect of inlet swirl on turbine efficiency and heat transfer by means of coupled and uncoupled\(^{10}\) CTI simulations. An efficiency reduction of more than 2% by swirl was found with a significant influence of swirl direction and a rather small influence of clocking. The sensitivity to swirl direction was strongly increased in transient simulations, compared to steady ones. A large influence of $\overline{\mathrm{Tu}}$ on efficiency was identified. Changing 0D to 1D and finally 2D inlet BCs steadily decreased the predicted efficiency. A maximum of HTC was identified in the rear part of vane passage, its circumferential location depended on swirl orientation.

Hilgert et al. [73] conducted SAS simulations and compared them with aerodynamic measurements and heat transfer results on the stator hub at different axial planes and two cooling MFRs. SAS turbulence modelling showed to be superior over the Shear Stress Transport (SST) model in resolving the swirl induced aerodynamics, especially the distribution of swirl angle at turbine inlet. Nu distributions on the stator hub could be qualitatively reproduced by the simulations.

The effects of inlet swirl on vane aerodynamics have been investigated in several other linear and annular test rigs, as well as real engine configurations. Giller and Schiffer [54] investigated the effect of inlet swirl on flow and film cooling of a linear cascade at different swirl numbers and distances between swirl generators and vanes. They identified an alteration of classical passage secondary flows at passage exit, a tilt of the stagnation line leading to an off-design operation of film cooling and a turnover of the SS-LE cooling film to the PS.

Schmid and Schiffer [158] conducted numerical simulations of the same cascade.

---

\(^{10}\) According to Fig. 1.3
examining the effect of inlet swirl strength and clocking. It was found, that inlet swirl increased heat transfer upstream of the vanes due to elevated turbulence and decreased HTC in the vane passage. Since the generated swirl cores with equal sense of rotation quickly merged after entering the passage, only a small effect of clocking was observed.

Investigations at the Oxford Turbine Research Facility: Qureshi et al. [140] investigated the influence of inlet swirl on heat transfer to the MT1 vanes at LE clocking. Peak swirl angles were at $\pm 40^\circ$ ($S = 0.7$ in the combustor simulator upstream). They discussed swirl induced vorticity at NGV exit and observed upwash and downwash on the vane surfaces in line with the rotation of swirl, causing accumulation of boundary layer fluid at the respective vane/end wall corners. Streamline redistribution was most pronounced on the vane PS exposed to swirl. The local thinning of the boundary layer caused enhanced heat transfer ($\Delta \text{Nu}$ was over 100\%). HTC at the end walls was locally enhanced by up to 25%.

In a companion paper Qureshi et al. [138] also examined rotor heat transfer. An increase in rotor relative swirl angle by $4^\circ$ in the top part (increasing rotor losses) and a decrease in the lower part by $4^\circ$ (decreasing rotor losses) due to swirl are reported. Rotor casing temperature and HTC were almost unaffected by swirl, HTC on the blades was slightly increased. Compared to the vanes, the effect of swirl on rotor blade HTC was small (max. $\Delta \text{Nu}$ was 40\% at the PS tip due to highest incidence there.).

A similar experimental and computational study on the same rig was conducted by Beard et al. [15] focussing on the efficiency deficit by swirl which was determined to be about 1\%. Losses occurred mostly in the stator. Turbine capacity changed by about 2\% which was underpredicted by CFD. A 1D throughflow analysis was found to agree well with 3D CFD indicating that losses in the $x-\theta$ plane are relevant for efficiency losses.

Investigations at the EPFL Cascade: Another test case, that has recently been used for numerical CTI studies, is a cascade rig of transonic high pressure vanes at École Polytechnique Fédérale de Lausanne (EPFL) equipped with a realistic film-cooling design (showerhead cooling at the LE and fan shapes holes on PS and SS). The vane has been investigated experimentally in the European project TATEF2 by Jonsson and Ott [93]. Insinna et al. [81] numerically investigated the effect of inlet swirl on heat transfer in the EPFL cascade at two clocking positions, using the $k_T$-$k_L$-$\omega$ turbulence model. Three major effects were observed, a shift of the stagnation line at the LE, convection of showerhead coolant to the hub, and a deterioration of film cooling on the vanes.
Other Investigations: The effect of inlet swirl on efficiency in a 1.5 stage research turbine was investigated by Pyliouras et al. [137] and compared to real engine simulations of the Engine 3E (E3E) by Rolls-Royce Deutschland (RRD). The stage efficiency deficit was 3.1% in the rig and 1.5% in the real engine. The effect of clocking was found to depend largely on the ratio of combustor to turbine sectors. In the E3E, realistic inlet conditions led to an increase of 400 K in vane surface temperature and 100 K in rotor tip temperature. Different authors have used numerical optimisation of the vane and rotor geometry in order to reduce aerodynamic loss from inlet swirl and recover turbine efficiency. HPT optimisations have been reported by Shih and Lin [166] and Shahpar and Caloni [163, 164] who recovered 1% in turbine efficiency. An integrated combustor vane concept for can combustors, in which NGVs are removed and flow turning is achieved by vanes that extend the combustor walls, is proposed by Rosic et al. [147] and aerodynamically and thermally optimised by Aslanidou et al. [4] and Jacobi and Rosic [85, 86, 87].

Inlet Swirl and Hot Streaks

Development of Experimental Facilities: As pointed out in Section 2.1.3, both swirl and hot spots can be found simultaneously at combustor exit and, in general, an interaction must be expected. This motivated the design of combustor simulators, allowing for an experimental investigation of this interaction. The design and testing of a non-reacting combustor simulator with swirl and temperature distortion are described by Hall et al. [64] and Hall and Povey [63]. They report the challenge of generating a representative swirl field in a non-reactive swirl simulator that lacks the stabilisation from heat release (cf. Section 2.1.2). The emerging PVC structure could not be suppressed by the injection of an axial jet. However, even in the presence of PVC the target outlet distributions could be achieved.

Numerical Investigations using the MT1 Turbine: Insinna et al. [79] conducted a numerical CHT study with realistic inlet conditions of the cooled MT1 stator row. Swirl led to overturning in the centre of the passage exit, altered vane loading and increased losses. Effects of inlet non-uniformities on the efficiency of the cooling system were negligible. Mass flow through the holes varied by 2% between uniform and non-uniform inlet conditions. Coolant density ratio DR was mainly affected by the inlet temperature profile whereas blowing ratio BR was additionally influenced by the swirl induced stagnation line shift. The presence of inlet swirl was found to smear out the effect of clocking on vane average temperature. Khanal et al. [95] conducted numerical studies on the interaction of swirl and hot streaks in the MT1. LE aligned swirl was split in two co-rotating pairs, twisting the hot streak and causing upwash and downwash on the vane surfaces. The induced
flows resulted in a selective radial transport of hot fluid to the hub and casing. Passage-aligned swirl stretched the hot spot which was explained by low momentum fluid in the boundary layers being more responsive to overall pressure gradients. Radial positions of pressure loss were found to depend on swirl clocking position and direction. Rotor loading (mostly on the PS) was affected mainly by swirl direction. A non-linear influence of swirl and hot streaks on rotor heat transfer was reported, i.e., the results from separate analyses cannot be superimposed.

**Numerical Investigations using the EPFL Cascade:** A similar study was conducted by Insinna et al. [82] on the EPFL cascade. Effects of inlet swirl and temperature non-uniformity on the cooling system were investigated by means of CHT simulations. Uniform temperature at the inlet resulted in hotter end walls (in agreement with Qureshi et al. [139]) whereas the realistic inlet profile led to overheating in the rear part of one vane due to a hot streak and reduction of cooling effectiveness from swirl. Consequently, non-uniform inlet conditions caused higher temperature peaks on the vanes. The averaged temperature, however, was higher with uniform inflow because about 30% of the heat flux was conducted from the end walls to the vanes. Realistic inflow strongly reduced this heat conduction from the end walls because the end wall temperature decreased, but only slightly increased convective transport through PS and SS. Heat flux and mass flow through the different rows of cooling holes were significantly affected by the realistic inlet profile. Both were mostly increased in the lower part and decreased in the upper part of the vane. Griffini et al. [58] studied clocking effects of realistic non-uniform inlet conditions (from measurements at the OTRF by Qureshi et al. [138]) on the EPFL cascade by means of CHT. It showed, that LE clocking of swirl is worse in terms of thermal load of the vane exposed to swirl. Its temperature was 10% higher because the cooling film was disturbed by inlet swirl. The hot spot was found to impinge the hub behind the vane exposed to swirl and film-cooling effectiveness was reduced in this region. As in the previously described study, higher temperatures at the combustor liner with uniform inlet led to higher averaged vane temperatures.

**Investigations at the Trisector Rig:** Most recently, the Trisector rig was developed at the University of Florence (UNIFI) as part of the European project FACTOR as preliminary test rig for a combustor simulator to be implemented into the NG-Turb test rig at Deutsches Zentrum für Luft- und Raumfahrt, Göttingen. The Trisector rig is an unscaled, non-reactive (but heated), 54° annular rig with three swirlers generating a lean-burn representative swirl field and a row of five partially film-cooled NGVs, designed for inlet swirl. The swirlers are shielded by ducts to preserve the swirl up to NGV inlet. Flows from liner effusion-cooling systems in front of the combustor exit plane can be used to generate an inhomogeneous temperature profile.
The design of the combustor simulator was based on LES by Koupper et al. [99]. Flowfield and temperature measurements behind the swirler (without NGVs), by means of probe measurements and particle image velocimetry are reported by Bacci et al. [5]. They report that ducts around the swirlers help to prevent mixing and preserve the high swirl up to the turbine inlet. Iso-thermal turbulence field measurements, by means of hot-wire anemometry with split-fiber probes, have been conducted by Bacci et al. [6]. The maximum turbulence intensity at the inlet to the turbine in a lean burn-like configuration amounted to 25 to 28%. Values were lower for unducted configurations and decayed at a downstream measurement location. Integral turbulent time scales from these measurements were compared to LES by Koupper et al. [101], who also proposed a method to account for different durations of data acquisition in experiment and CFD.

More recently, the flow and temperature migration, with NGVs installed, were measured by Bacci et al. [8]. They found that swirling inflow is mainly convected to a single passage where a strong non-uniformity at outlet results from an accumulation of the hot flow at the hub. Other passages showed rather uniform exit temperature. Bacci et al. [7] further investigated the film cooling effectiveness on the NGVs at different relative clocking positions by means of pressure sensitive paint measurements. They identified an influence of the relative swirl clocking position due to shifts in the stagnation line. Inlet swirl generated an upwash of PS cooling and influenced LE cooling by its low pressure core.

Numerical simulations of the rig were conducted by different authors. Andreini et al. [2] compared SAS with simpler EVMs for the turbulent closure (in the commercial solvers ANSYS® CFX® and FLUENT®) of an unducted swirler (without NGVs installed). The simple EVMs were found to overpredict the extent of the recirculation zone and the position and magnitude of axial velocity peaks. Also, a strong influence of the periodicity assumption on the temperature profile was identified (lateral walls caused a non-periodicity in the experiments). In a second publication by Andreini et al. [3], investigating the same set-up, SAS turbulence closure showed noticeable improvements in the prediction of the recirculation zone and the hot spot at the combustor outlet.

The effect of turbulence modelling on the prediction of the flow on the interface was evaluated by Cubeda et al. [38], who compared the SAS against the SST-k-ω model. They state that time-averaged aerodynamics at combustor exit can be sufficiently predicted by SST computations but turbulence (and thus wall temperatures and hot streak diffusion) cannot. Discrepancies between SST and SAS were at the order of 0.1 in LOTDF at plane 40 and 100 K in adiabatic wall temperature.

Koupper et al. [100] conducted LES simulations of the FACTOR set-up, with NGVs installed, at two clocking positions of the hot spot. In LE aligned position, it considerably increased the heating of the vane but also mixed out more towards the end.
of the passage, compared to a passage clocking position. Also, a marked effect of clocking on the propagation of combustor coolant through the nozzle was observed. An upstream effect of the vanes did not alter the temperature field but an effect on flow angles was noticed up to 30% \( \text{ch}_{\text{ax}} \) upstream of the LE. Potential effects on turbulence depended on the clocking position.

Coupled and decoupled LES of the configuration including a rotor row were conducted by Duchaine et al. [46]. Although both showed similar aerodynamic effects, differences in the nozzle wall temperature distribution were observed in the decoupled simulations with steady inlet conditions due to the missing coherent structures exiting the combustor.

2.4.3 Implications for this Work

The following implications on the development of a combustor exit flow model and the CFD simulations conducted in this work can be derived from the literature study:

- **Comparability**: The different investigated turbines and inflow conditions are heterogeneous and comparability between the cases is limited to the observation of general effects (→ a dynamic method, flexible with respect to combustor and turbine design, is developed and applied to different cases).

- **Mean inlet temperature**: Hot spots are relevant for NGV and rotor heat load but affect aerodynamic effects only in the rotor (→ results from iso-thermal investigations are transferable). Low temperatures from liner coolant at turbine inlet have a significant influence on predicted heat loads (→ hot spots and the radial temperature profile are included in the flow model).

- **Mean inlet swirl**: The effect of inlet swirl is mostly confined to the NGV (→ effects are analysed in steady-state simulations of the NGV). It deteriorates film cooling and may induce additional secondary flows under certain conditions (→ formation of swirl-induced secondary flows is investigated).

- **Interaction of hot spot and swirl**: In general, the thermal and swirl field interact and their aero-thermal effects cannot be investigated in isolation (→ a combined UQ analysis is conducted). The clocking position of hot spots and swirl, relative to the vanes, are the most important characteristics of HPT inlet conditions (→ clocking effects are investigated in the UQ analysis).

- **Turbulent fluctuations at inlet**: A correct estimation of the inlet turbulence level is important for the correct prediction of hot spot diffusion and efficiency. Turbulent inlet conditions can be predicted correctly only by scale-resolving turbulence models (→ inlet turbulence is obtained from experiments or SAS predictions, if possible.).
• **Limitations of CFD:** Steady CFD cannot predict the thermal conditions of the rotor (→ the rotor thermal field is not analysed in steady CFD) and quantify absolute values of efficiency correctly (→ only efficiency differences are investigated). Realistic metal temperatures can only be predicted by means of CHT simulations taking heat conduction to and from the end walls into account (→ predicted metal temperatures are not used for an evaluation of turbine life). Conditions of the external flow influence the operating conditions of coolant ejection (→ inner vane cavities are resolved in CFD, if possible). In general, a decoupled turbine simulation cannot reproduce time-resolved turbulent structures from a coupled simulation (→ only time-averaged flow fields are analysed).
3 Methods Review
Computational Fluid Dynamics and Uncertainty Quantification

3.1 Fundamentals of Computational Fluid Dynamics

The following passages give an overview of turbomachinery CFD in a typical industrial context using RANS turbulence modelling and finite volume discretisation, as far as they are relevant for the methods applied in this work. For a more extensive review of the topic, the books by Hirsch [74], Ferziger and Peric [50] and Schäfer [155] are recommended. An overview on fundamentals of turbulent flows is given. An extensive discussion can be found in Pope [134]. The mathematical modelling as well as the discretisation of the respective equations is described. Furthermore, sources of numerical errors are classified.

3.1.1 Conservation Equations

Fluid flow problems are governed by the conservation of mass, momentum, energy and species. For each of the conserved quantities $\Phi$, a transport equation

$$
\frac{\partial}{\partial t} (\rho \Phi) + \frac{\partial}{\partial x_i} (\rho u_i \Phi) = \frac{\partial}{\partial x_i} \left( \Gamma_{\Phi} \frac{\partial \Phi}{\partial x_i} \right) + S_{\Phi}
$$

(3.1)

can be derived by balancing the fluxes through the surfaces of and the sources and sinks in a finite volume. The resulting equations for a single-phase flow are referred to as continuity equation, momentum equation and energy equation, respectively. The derivation can be found in a number of text books, such as Bird et al. [19], and is not reproduced here. The conservation equations hold for all types of fluids (materials that do not support shear stress). Hence, the properties of the simulated fluid need to be specified. In turbine CFD the mixture of burned fuel and air is commonly assumed to be a Newtonian fluid, obeying Stokes hypothesis, as well as a thermally and calorically perfect gas. Heat flux is assumed to obey Fourier’s Law. This set of equations, accompanied by a set of boundary conditions, yields a complete mathematical model for the solution of flow problems. In practical applications, however, the effects of turbulent fluctuations are modelled to increase numerical efficiency.
3.1.2 Turbulent Dynamics and Scales

Flows in many technical applications are turbulent. Although there is no universal definition of turbulence, it is commonly described to be an unsteady, irregular, random and chaotic motion of the flow. The transition of a laminar to a turbulent flow is determined by the flow Reynolds number

$$\text{Re} := \frac{\rho u L_{\text{ref}}}{\mu} \equiv \frac{\text{inertia}}{\text{viscous stress}}$$  (3.2)

which relates the inertial to the viscous forces in the flow. Coherent turbulent structures are often symbolically represented by vortices, referred to as eddies, which are characterised by an individual length and time scale. A fundamental concept, the energy cascade (Fig. 3.1), explains the emergence and decay of these structures. Turbulent eddies emerge from shear in the mean flow, i.e., kinetic energy $E_{\text{kin}}$ of the mean flow is transferred to turbulent kinetic energy $k$. Eddies produced by mean shear (at the size of a typical length of the fluid domain) are large in relation to the spectrum of turbulent scales. These large, energy containing eddies decay to ever smaller and smaller ones, ranging over several scales in size. The turbulent decay is assumed to be energy-conserving, i.e., the rate at which turbulence energy is produced is approximately equal to the rate of its dissipation $\epsilon$. The range of decaying eddies, that are large enough not to be affected by viscous forces, is called inertial range. When the eddies finally reach the Kolmogorov scales, determined by the fluids kinematic viscosity $\nu$ and the energy transfer rate of the turbulent cascade $\epsilon$, viscous forces become dominant and they are dissipated by heat. According to Kolmogorov [98], at these smallest scales turbulence is isotropic and statistically universal, i.e., independent of the large eddies it emerged from.

**Figure 3.1:** The turbulent energy cascade: Kinetic energy is transformed to turbulent kinetic energy of large coherent eddies to successively smaller ones until it is dissipated into heat at the Kolmogorov scale.
Numerical solution methods for turbulent flows are distinguished by their ability to resolve the turbulent energy spectrum. Direct Numerical Simulations (DNS) solve the Navier-Stokes equations directly on all scales of the spectrum without any modelling of turbulence which requires an enormous numerical effort, scaling with the third power of the turbulent Reynolds number. The principle of Large Eddy Simulations is to apply a filtering of certain width to the turbulent spectrum. All turbulent structures above this filter width are resolved, whereas the filtered turbulent sub-grid stresses must be provided by a model. The Reynolds-Averaged Navier-Stokes approach does not resolve any of the turbulent structures but relies entirely on turbulence models. Due to its enormous computational efficiency RANS is currently the industrial standard for the simulation of most turbulent flows. However, the application of LES to technically relevant problems becomes increasingly affordable.

3.1.3 Reynolds-Averaged Navier-Stokes Turbulence Models

The RANS Equations

In many engineering design problems the knowledge of the time mean of a flow field is sufficient, i.e., one is not interested in the turbulent fluctuations. RANS models therefore rely on the concept of Reynolds averaging, expressing the flow quantities \( \Phi(x,t) \) as the sum of a time mean \( \overline{\Phi}(x) \) and an unsteady fluctuation \( \Phi'(x,t) \),

\[
\Phi(x,t) = \overline{\Phi}(x) + \Phi'(x,t), \quad \text{where} \quad \overline{\Phi}(x) := \lim_{T \to \infty} \frac{1}{T} \int_0^T \Phi(x,t) \, dt.
\] (3.3)

Conveniently, the time mean of a fluctuation is zero, i.e., \( \overline{\Phi}' = 0 \). Insertion of the Reynolds-averaged quantities into the conservation equation (3.1) and subsequent time averaging thus yields a set of Reynolds-averaged transport equations for \( \overline{\Phi} \) that resembles those of the instantaneous quantities \( \Phi \) in Eqn. (3.1). In the averaged transport equations of momentum a single unknown term \( \overline{u_i u_j} \) occurs representing the turbulent stresses or Reynolds stresses. This term, however, cannot be derived from known quantities because higher order transport equations for the unknown tensor contain ever more unknowns which is known as the closure problem. Therefore, provision of the turbulent stress tensor, using only known quantities, is the purpose of RANS turbulence models.

The energy content in the turbulent velocity fluctuations is given by the turbulent kinetic energy \( k \) which is the trace of the Reynolds stress tensor. A more tangible

---

1 Compressible flows are, strictly speaking, not Reynolds-averaged as in Eqn. (3.3) but commonly Favre-averaged, defining a mass-weighted mean \( \overline{\Phi} = \overline{\rho \Phi / \rho} \) in order to implicitly account for density fluctuations by the averaging procedure (CFX® [29]).

2 Similar terms appear in the conservation equation of energy and species which can be modelled by the same approaches described here for the momentum equations.
quantity is obtained by normalising the root of $k$ with the local mean flow velocity $\bar{u}$ yielding the turbulence intensity

$$ Tu := \frac{\sqrt{2/3} \, k}{\bar{u}}, \text{ where } k = \frac{u'_{i}u'_{j}}{2}. \quad (3.4) $$

### Eddy Viscosity Models

Eddy viscosity models (EVMs) rely on a hypothesis by Boussinesq [23], assuming a proportionality of the Reynolds stresses to the shear rate of the mean velocity $U_{ij} = \left( \frac{\partial u_{i}}{\partial x_{j}} + \frac{\partial u_{j}}{\partial x_{i}} \right)$ for modelling $u'_{i}u'_{j}$. In analogy to Newton’s definition of molecular viscosity $\nu$ the proportionality constant is referred to as turbulent or eddy viscosity $\nu_{t}$ and

$$ u'_{i}u'_{j} - \frac{2}{3}\delta_{ij}k = \nu_{t}U_{ij} \text{ with } \nu_{t} \propto L^2 \mathcal{T}^{-1}. \quad (3.5) $$

Turbulent motions are thus assumed to act as viscous stresses $\nu_{t}$ which effectively increase molecular viscosity $\nu$ such that $\nu_{\text{eff}} = \nu + \nu_{t}$. The eddy viscosity $\nu_{t}$ is computed locally from a turbulent length scale $L$ and time scale $\mathcal{T}$ according to the dimensions of viscosity. The two scales $L$ and $\mathcal{T}$ depend on the respective flow problem and must be provided by solving additional transport equations in two-equation models. The time scale $\mathcal{T}$ can be provided comparatively easily (assuming, a respective length scale $L$ is known) by solving a transport equation for turbulent kinetic energy $k$ of dimension $L^2 \mathcal{T}^{-2}$. The equation is derived by multiplying the transport equation for a velocity fluctuation $u'_{i}$ with $u'_{i}$ itself and averaging the resulting equation, such that

$$ \frac{Dk}{Dt} = u'_{i}u'_{j} \frac{\partial u_{i}}{\partial x_{j}} - \nu \frac{\partial u'_{i}}{\partial x_{j}}^2 + \frac{1}{\rho} \frac{\partial}{\partial x_{j}} \left( \mu \frac{\partial k}{\partial x_{j}} - \rho k' u'_{j} - p' u'_{i} \delta_{ij} \right). \quad (3.6) $$

The three terms on the right side of this equation reflect the dynamics of turbulence discussed in Section 3.1.2. Velocity gradients $\frac{\partial u_{i}}{\partial x_{j}}$ in the source term $\mathcal{P}$ transfer energy from the main flow into turbulent kinetic energy $k$, on the other hand $k$ is dissipated into heat, at the Kolmogorov scale, in the sink $\epsilon$. The diffusion term acts neither as source nor sink of $k$ but merely redistributes it. Eqn. (3.6) is exact, but for the solution by a CFD code, the terms are modelled. The production term $\mathcal{P}$ is closed by the Boussinesq hypothesis, Eq. (3.5), and the dissipation $\epsilon$ is usually

---

3 Simpler algebraic or one-equation models provide either both or one of the scales from algebraic equations without solving additional transport equations.
provided by a second transport equation. The diffusion term is modelled by a gradient diffusion approach\(^4\), based on the eddy diffusivity hypothesis

\[
\overline{\phi' u_j'} = -\frac{\nu_t}{\sigma_\phi} \frac{\partial \overline{\phi}}{\partial x_j},
\]

i.e., diffusion of fluctuations \(\phi'\) is assumed to be driven by the gradient of the mean quantity \(\overline{\phi}\) with a diffusivity controlled by a parameter \(\sigma_\phi\). In the \(k\)-equation \(\phi \equiv k\) (pressure transport is often neglected). In the modelling of the analogue term in the turbulent energy equation, the same method is used by analogy of heat and momentum transport and \(\phi \equiv T\). In this case, the coefficient \(\sigma_\phi \equiv \text{Pr}_t\) is referred to as the turbulent Prandtl number, in analogy to the molecular Prandtl number \(\text{Pr} := \nu c_p / k\) (where \(k\) is the thermal conductivity).

Whereas the \(k\)-equation for the provision of a time scale \(T\) can be modelled with relatively few assumptions, providing a turbulent length scale \(L\) is much more challenging. The most common scale providing variable is the turbulent dissipation \(\epsilon\) (as in the well known \(k-\epsilon\)-model by Launder and Sharma \([106]\)). A transport equation for \(\epsilon\) can be derived analytically, however, nearly all terms of this equation need to be modelled. Also, using the exact \(\epsilon\)-equation can be problematic, as it describes turbulent dissipation on the Kolmogorov scales, whereas the quantity needed for provision of a turbulent length scale \(L\) actually describes energy transfer from the largest energy-containing eddies on the other side of the turbulent spectrum (cf. Fig. 3.1) and the two rates of energy transfer are not necessarily equal (Hanjalic \([66]\)). Hence, the \(\epsilon\)-equation is modelled heuristically and composed similar to the \(k\)-equation with a production, dissipation and diffusion term. Due to the large number of assumptions, the error in the computation of turbulent length scales \(L\) is, in general, much larger than that of time scales \(T\) in two-equation models (cf. Section 3.4).

The major advantage of EVMs is their robustness which is especially important for the application in turbomachinery CFD. On the other hand, EVMs face some major deficits in physical modelling (Hanjalic \([66]\)) which are mainly a consequence of the Boussinesq hypothesis. It can be shown that Eqn. (3.5) is a linearised form of a more general, non-linear relation for eddy viscosity (Pope \([134]\)). Due to the linearisation, the dependency of \(\nu_t\) on the rotation tensor\(^5\) \(\Omega_{ij}\) of the velocity field is lost. This causes deficits in the description of streamline curvature and rotation by linear EVMs

\(^4\) The rationale behind this approach can be thought of similar to Fourier’s Law of heat condition: Diffusion acts against the gradient of the driving concentration, just as heat \(\dot{q}\) flows against a driving temperature gradient \(\partial T / \partial x\).

\(^5\) Any matrix can be decomposed into a symmetric and an anti-symmetric matrix. Therefore, the Jacobian of a velocity field \(\partial u_i / \partial x_j\) can be decomposed into a symmetric tensor \(U_{ij}\) describing the strain rate and an anti-symmetric tensor \(\Omega_{ij} = \frac{1}{2}(\partial u_i / \partial x_j - \partial u_j / \partial x)\) describing rotation.
which is a major deficit in the context of turbomachinery and swirling flows. Also, turbulent anisotropy and production driven by normal stresses in the main flow cannot sufficiently be described (Hanjalic [66]). It is therefore important to assess modelling errors with respect to turbulence by comparing data with experiments or simulations with more advanced turbulence models.

Reynolds Stress Models

Reynolds stress models are, like EVMs, a class of RANS models but at a higher level of complexity. Unlike EVMs, they do not rely on an eddy viscosity $\nu_t$ as defined by the Boussinesq hypothesis (3.5) and thus avoid its previously mentioned disadvantages. Instead, a higher order transport equation for the Reynolds stress tensor is derived from which $\overline{u_i'u_j'}$ is calculated directly next to a scale providing variable, such as $\epsilon$. This equation contains several unclosed, higher order terms, represented symbolically by tensors in Eqn. (3.8), which need to be modelled.

In the literature, turbulent diffusion $D_{ij}$ is commonly modelled by a tensorial, anisotropic gradient diffusion approach as proposed by Daly and Harlow [40]. In this work, however, the isotropic formulation in Eqn. (3.7) is used, which is the default setting in CFX® [29], in order to improve stability of the numerical solution. Assuming isotropy of dissipation $\epsilon$ on small scales, in agreement with Kolmogorov’s hypothesis, allows to compute the dissipation tensor $\epsilon_{ij}$ in scalar form, i.e., $\epsilon_{ij} = \frac{2}{3} \epsilon \delta_{ij}$.

The major challenge, and at the same time the main difference between RSMs, is the modelling of the pressure strain correlation $\Pi_{ij}$. This term does not produce nor dissipate turbulent kinetic energy but merely redistributes it in the flow domain and thus influences turbulent anisotropy. In order to model the term, a transport equation for pressure fluctuations $p'$ is derived and differentiated. The solution of the resulting Poisson equation is multiplied with the shear rate and time averaged, yielding the wanted expression for $\Pi_{ij}$. It contains two volume integrals of double and triple correlations of the unknown velocity fluctuations which must be modelled. Due to their physical properties, the terms are referred to as rapid term $\Pi_{ij}^{ra}$ and return term $\Pi_{ij}^{re}$. The $\omega$-based RSMs in CFX® [29], such as the RSM-BSL model used in this work, model the return of anisotropic turbulence to an isotropic state proportional to the anisotropic tensor $b_{ij}$ as defined in Eqn. (3.9). The rapid term is modelled similar to the formulation of the quasi-isotropic Launder-Reeece-Rodi (LRR) model (Launder et al. [107]). The well-known SSG model by Speziale et al.
Explicit algebraic Reynolds stress models (EARSMs), are a simpler, more robust class of RSMs. Reynolds stresses $u'_i u'_j$ are modelled based on the anisotropic tensor $b_{ij}$

$$
\overline{u'_i u'_j} = k(b_{ij} + 2/3\delta_{ij})
$$

which is calculated from algebraic relations and thus more robust than ordinary RSMs based on differential equations. EARSMs can therefore be regarded as compromise between the robustness of EVMs and the physical properties of RSMs. The formulation by Wallin and Johansson [178] is implemented in CFX® [29].

Wall Treatment

Flows through jet engines are internal. Hence, the bounding walls have a significant influence on the flow. According to Prandtl, characteristics of turbulent flows in the proximity of a wall change with non-dimensional wall distance ($\text{Spurk [169]}$)

$$
y^+ := \frac{yu^+}{\nu}, \quad \text{where} \quad u^+ = \sqrt{\frac{\tau_w}{\rho}}.
$$

In the viscous sublayer, close to the wall ($y^+ < 5$), turbulent stresses are much smaller than viscous stresses. The influence of turbulence grows with increasing wall distance and becomes dominant in the area of the law of the wall ($y^+ > 30$). In order to be able to study effects of the wall boundary layer as well as wall heat transfer to and from solid walls, the flow must be resolved down to the viscous sublayer by the numerical grid ($\text{low-Re}^6$ approach).

The physical phenomena at the walls must be reproduced by turbulence models. However, all of the aforementioned $\epsilon$-based models struggle to produce the high viscosity in this region (Hanjalic [66]). A common solution is to introduce damping functions $f_\mu(y)$ adjusting the eddy viscosity in the vicinity of walls. As an alternative, the use of the specific dissipation rate $\omega = \epsilon/k$ instead of $\epsilon$ as scale providing variable has been proposed by Wilcox [185]. His $k$-$\omega$-model does not require a near wall modification. Using $\omega$, however, is problematic for the specification of free stream boundary conditions (Hanjalic [66]). The disadvantages of both models led to the introduction of the Shear Stress Transport model by Menter [117]. The SST-$k$-$\omega$ model uses the $\omega$-formulation close to walls and the

---

6 The numerical grid is iteratively adjusted, using $u^+$ from CFD, until $y^+ \approx 1$. An alternative method ($\text{high-Re}$ approach) is to resolve the flow only down to $y^+ \approx 30$ and impose boundary conditions in accordance with the law of the wall that models the effect of the boundary layer ($\text{wall functions}$).
formulation in the main flow. The respective coefficients $C$ of the model are determined by a blending function $F_1$:

$$C_{\text{SST}} = F_1 C_\omega + (1 - F_1) C_\epsilon.$$  (3.11)

Due to its robustness and applicability to a range of problems, SST-k-$\omega$ has become the standard EVM in industrial application with respect to turbomachinery CFD. The $\omega$-based RSMs used in this work (all except SSG) make use of a similar blending technique in order to allow the resolution of wall boundary layers.

A problem of many two-equation models is the overestimation of $\nu_L$ as $L$ corresponds to the local thickness of the turbulent shear layer which prevents the resolution of turbulent decay appropriately. The Scale Adaptive Simulation (SAS) model by Menter and Egorov [118] is a two-equation RANS model, targeted to overcome this weakness. It is based on an exact transport equation for $kL$ by Rotta but features a term in the expansion of production which was originally neglected by Rotta. Since $\sqrt{kL}$ is used as scale determining variable, SAS is much better able to yield realistic turbulent length scales $L$ than classical two-equation models. The model is convenient in practical application because it allows blending an “LES-like” mode into RANS, based on the local refinement of the mesh. The numerical effort, however, is still well above that of steady RANS calculations.

3.1.4 Discretisation and Solution of Conservation Equations

Discretisation of Conservation Equations

There is no known analytical solution of the full Navier-Stokes Equations which is why flow problems are solved numerically by means of CFD. The numerical solution is an approximation of the continuous physical fields of flow quantities $\Phi$ provided at discrete points in space and time.

There are different methods for spatial discretisation of which the Finite Volume Method (FVM) is the common approach for fluid flow problems. Its main advantage is the inherent conservativeness of the discretisation. Summation of the discretised integral equations (3.12) over the entire domain yields a single conservative equation ([Ferziger and Peric 50]) which shows that no sources or sinks of the conserved quantities within the domain arise from the discretisation.

The principle of the FVM is a division of the physical domain into several finite volumes (cells) that the transport equations (3.1) apply for. These continuous equations are discretised in order to express all terms as functions of quantities $\Phi_p$ at discrete points (usually the centres of the cells). In order to formulate conservative equations for each cell, the differential equations (3.1) are integrated over volume $V$. According to the divergence theorem, the volume integral of the divergence of a
tensor field is equal to the outward flux of this field. Application of this rule yields the integral form of equations (3.1)

\[
\int _{V} \frac{\partial}{\partial t} (\rho \Phi) dV + \int _{S} \rho u_i \Phi \cdot n_i dS = \int _{S} \Gamma _{\Phi} \frac{\partial \Phi}{\partial x_i} \cdot n_i dS + \int _{V} S_{\Phi} dV, \tag{3.12}
\]

where the integrals of convective and diffusive terms have been transformed to integrals over surface \(S\). The integrals in these equations must be approximated by discrete expressions for a numerical solution. The approximation of the volume integration of the source terms \(\int _{V} S_{\Phi} dV\) is relatively simple and can be done by the second order midpoint rule or other rules of higher order (Ferziger and Peric [50]). The surface integrals \(\int _{S} f dS\) of convective and diffusive fluxes are approximated by discrete values \(\Phi _{S}\) and gradients \(\frac{\partial \Phi _{S}}{\partial x}\) on the surface of the cells using similar schemes. But, since only the values \(\Phi _{p}\) in the cell centre are known, \(\Phi _{S}\) and \(\frac{\partial \Phi _{S}}{\partial x}\) must first be expressed in terms of these known values \(\Phi _{p}\). The differencing schemes available for these computations are distinguished by the order \(n\) of which their error scales with grid spacing \(\Delta\). Increasing the discretisation order \(n\) is a trade-off between increasing accuracy and decreasing computational efficiency and stability. The second order Central Differencing Scheme (CDS) is the current industrial standard with regard to CFD in turbomachinery and computes \(\Phi _{S}\) and \(\frac{\partial \Phi _{S}}{\partial x}\) from linear interpolations of the midpoints \(\Phi _{p}\) of the neighbouring cells. The first order Upwind Differencing Scheme (UDS) is computationally more efficient and stable but suffers from the occurrence of numerical diffusion\(^8\) which is introduced into the transport equations by the leading error terms of the Taylor series. Schemes of higher orders (above three) are rarely used in practical applications due to the high numerical effort. For the CFD results shown in this work, a flux blending approach is used which combines the first order UDS and second order CDS (Schäfer [155]) in order to increase robustness of the numerical solution process. A coefficient \(\Phi\) is then computed from

\[
\Phi = (1 - \beta) \Phi _{UDS} + \beta \Phi _{CDS} \quad \text{with} \quad 0 \leq \beta \leq 1, \tag{3.13}
\]

where \(\beta\) is adjusted locally by the solver in order to grant boundedness of the solution (CFX\(^\text{®}\) [29]).

Similar to spatial discretisation, the temporal derivative in Eqn. (3.12) can be discretised by schemes of different order. The computation of the next time step

---

\(^7\) The order of the discretisation error can be determined by comparing the scheme’s approximation with a truncated Taylor series about \(\Phi _{S}\).

\(^8\) That is, UDS “produces” artificial, numerical diffusion that acts just like physical diffusion and thus affects the numerical solution.

3.1 Fundamentals of Computational Fluid Dynamics
\( t + \Delta t \) can be done \textit{explicitly}, based on information \( \Phi(t) \) of the current time step, or \textit{implicitly}, based on the next time step \( \Phi(t + \Delta t) \), such that

\[
\Phi(t + \Delta t) = \begin{cases} 
\Phi(t) + f(t, \Phi(t))\Delta t & \text{(explicit)} \\
\Phi(t) + f(t + \Delta t, \Phi(t + \Delta t))\Delta t & \text{(implicit)}
\end{cases}
\]

(3.14)

The time step \( \Delta t \) of explicit schemes is limited by the \textit{CFL condition} (Courant et al. [37]), in order to grant stability of the numerical method, whereas implicit schemes remain stable with much larger time steps.

**Numerical Grids**

The distribution of mesh points, that the discrete numerical solution \( \Phi_p \) is computed on, in physical space (the generation of a computational \textit{mesh}) is one of the major practical challenges in the application of CFD. This applies especially for complex geometries with small features such as film cooling holes in turbines. In \textit{block structured} meshes, points are allocated in fixed relations to each other within each mesh block. Cells then typically have hexahedral shape and each point has the same number of neighbouring points which leads to a \textit{sparse} structure of the system matrix \( \mathbf{A} \) in Eqn. (B.1) of each block. This can be solved efficiently by specialised algorithms. However, generating a structured grid for complex geometries is extremely challenging and laborious because the small scale and large scale geometrical features must be meshed using the same topological arrangement of mesh blocks. This is, why \textit{unstructured} grids are generated for complex geometries as those in this work. These abandon the regular structure of grid points, resulting in a fully occupied matrix \( \mathbf{A} \), which decreases numerical efficiency. Since grid cells are typically tetrahedra, they cannot be aligned with a main flow direction which generally increases discretisation errors. On the other hand, unstructured meshes can be locally refined to better resolve large gradients in the flow field such as velocity wakes behind rotor blades. As tetrahedral cells with large aspect ratios in the boundary layer lead to numerical oscillations (Ferziger and Peric [50]), they are replaced by hexahedral cells in \textit{hybrid} meshes.

The quality of numerical meshes, influencing the solvers ability to quickly converge with small discretisation error, is quantified by different metrics. Cells must be orthogonal (inner \textit{cell angles} should be small), their sides should be of equal length (the cell \textit{aspect ratio} should be small) and adjacent cells should be of equal size (the cell \textit{element volume ratio}, EVR, should be small).

Evaluation of the discretised physical equations on the numerical grid, together with a set of boundary conditions, yield a closed system of equations. The methods applied for an efficient solution of this system are described in Appendix B.1.
3.1.5 Numerical Error

The numerical error is defined as the difference between the solution obtained by the CFD solver and the exact solution of the differential equations (3.1). It consists of two components (Schäfer [155]). The discretisation error is the difference between the exact solution of the original differential equations and their discretised approximation. The solution error is the difference between the exact and the actually obtained solution of the discretised approximations.

The solution error can be estimated indirectly by the residual which is the normalised difference between the left and right hand side of the system (B.1). The discretisation error is directly influenced by the order $n$ of the discretisation, as described in Section 3.1.4, as well as the grid spacing $\Delta$. It can be shown, that the error is proportional to $\Delta^n$. That is, the discretisation error decreases linearly with mesh refinement, if a first order discretisation scheme is used, quadratically, if a second order scheme is used, etc. A common method to estimate the discretisation error in a mesh study is therefore to run a simulation on multiple numerical meshes that are systematically, substantially and isotropically refined, until the solution is insensitive to further mesh refinement.

Apart from the numerical error there is a modelling error which accounts for the deviations of the real solution from that of the modelled differential equations. Among those models introducing a modelling error to the solution are the Navier-Stokes Equations themselves, fluid properties, turbulence and combustion modelling, the geometric model of the flow domain, boundary conditions, the assumptions of steadiness and periodicity. The modelling error can be estimated by comparison with experiments if the numerical error is known to be small.

3.2 Aspects of Simulating Flow through High Pressure Turbines

HPT simulations are challenging both in physical modelling as well as numerical discretisation and effort. This section is supposed to give an overview on the different aspects that must be considered.

Important questions with respect to the set-up of the geometrical model are if film cooling and purge flows are to be included in the model and, if so, where the respective domain inlets are to be placed. If internal cavities, transporting coolant in the blades, are not included, the physical domain must be cut off within the cooling holes and an inlet to the domain is assigned at each cut-off location. In steady simulations, this modelling approach is essentially a trade-off between model errors and numerical errors. Cutting off the cooling holes and placing domain inlets in the holes stabilises steady computations because transient interaction between cavities and main flow is prevented. Yet, a modelling error is introduced because flow conditions at these inlets are unknown and must be estimated. Also, the grid
generation method must provide the possibility to discretise film cooling holes, small geometrical features and purge flow cavities. If these features are to be included in the model, an unstructured grid must be used which generally decreases mesh quality. Furthermore, the geometrical model must be appropriately scaled to “hot” conditions accounting for thermal expansion and centrifugal forces.

Another important issue of the physical model is the definition of boundary conditions (cf. Fig. 3.2). Inlet conditions are scaled to thermodynamically comparable conditions, as discussed in Section 3.3.3. Inhomogeneities are either resolved, as in this work, or averaged out by using 1D profiles (Eqn. (2.4)) or 0D averages. Unsteadiness of inlet conditions is not resolved in decoupled HPT simulations. A typical assumption is that the flow in two adjacent blade passages is equal and the passage boundaries can be assumed periodic. This greatly increases computational efficiency but prohibits large scale sector interactions. The thermal boundary condition at the walls is important for balancing and thermal analyses. Walls are modelled adiabatic for aerodynamic simulations (as required by an isentropic efficiency definition, Eqn. (2.7)), whereas the computation of heat transfer at the walls requires a fixed wall temperature (cf. Section 2.3.1). Wall boundary layers are especially important for heat transfer problems and must be adequately resolved by the numerical grid. The interfaces between blade rows are important with regard to transient blade row interaction (TBR) which describes the transient impingement of wakes from an upstream onto a downstream blade row. The effect is not resolved in steady simulations using mixing planes but can be resolved by TBR models, as discussed in Appendix B.2, or scaling the geometry to integer blade counts. The assumption of steadiness furthermore suppresses the development of unsteady modes in the rotor purge flow cavities and the coolant films.

**Figure 3.2:** Implicit modelling assumptions due to the boundary conditions of turbine CFD
differences (at the order of 1000 K) require the modelling of temperature dependency of fluid properties. Also, components of unburned gas from the combustor may locally affect fluid properties in the turbine (the effect is not resolved in this work). Streamline curvature in the rotating domains and from residual combustor swirl as well as flow anisotropy challenge the modelling of turbulence with EVMs (cf. Section 3.1.3) and necessitate the use of RSMs. These, however, often lack the robustness required to interact with the various aforementioned effects.

### 3.3 Numerical Analysis of Combustor Turbine Interaction

#### 3.3.1 Shortcomings of Decoupling Combustor and Turbine

Combustor and turbine are two physically coupled sub-systems. It is therefore most reasonable to compute the flow through both components in one simulation. As outlined in Section 1.1.4, this approach still faces problems and has not yet advanced to an industrial standard. As a pragmatic workaround, the analysis of the sub-systems is decoupled by running two separate simulations and exchanging a boundary condition at the CTI interface (Fig. 1.3).

This workaround, however, comes with a number of problems as well. The position of the CTI interface is not arbitrary as it must be placed sufficiently far downstream of the combustor at a position where the reaction in the combustor is mostly completed, but also sufficiently far upstream of the turbine to minimise the influence of the NGV’s potential field on the traverse. Unsteady fluctuations from the combustor, if modelled in combustor CFD, cannot be resolved in the turbine simulation because a time-averaged flow field is set as turbine inlet condition. Furthermore, the decoupling causes a discontinuity in some of the flow quantities which is described in the next section.

#### 3.3.2 Discontinuity at the Combustor Turbine Interface

The inflow BC of HPT simulations is usually defined by distributions of total temperature, total pressure, flow angles and turbulent quantities

\[
\Phi_{BC} = [T_t, p_t, \alpha_{rad}, \alpha_{tan}, k, \epsilon].
\]  \hspace{1cm} (3.15)

Note, that imposing flow angles does only impose the direction of velocity vectors but not their magnitude \(u\). Since the distribution of velocity \(u\) in the combustor exit traverse is not prescribed by the HPT inlet BC, it may converge to an a priori unknown state in the CFD solution of the turbine flow. This “downstream simulation” (without an upstream combustor) thereby poses different restrictions to the velocity field than the decoupled “upstream simulation” of a combustor\(^9\) and the distribution

---

\(^9\) Differences are due to the modelling of the turbine geometry, the turbulence model, outflow conditions, unsteadiness in the upstream field, etc.
of \( u \) (and consequently the static quantities \( p, T, \rho \)) at the CTI interface is therefore per se different in the downstream and the upstream simulation. Thus, prescribing the (incomplete) set of upstream quantities \( \Phi_{BC} \), Eqn. (3.15), in the downstream CFD causes a discontinuity in all quantities which are not prescribed. In principle, other combinations of \( \Phi_{BC} \) are possible but these would be impractical with most solvers and shift the problem of discontinuity to other flow quantities.

In a practical application, the discontinuity in \( u \) is mainly relevant with respect to axial\(^{10}\) velocity \( u_{\text{ax}} \) due to its role in mass flow averaging. In order to allow for an estimation of the effect, the difference in \( u_{\text{ax}} \) is illustrated in Fig. 3.3 (a) for the LSTR\(^{11}\) which can be seen to reach up to 40% locally.

Averaging a quantity \( \Phi(r, \theta) \) on a flow interface is mathematically done by evaluating

\[
\Phi = \frac{\int \int \Phi(r, \theta) \xi(r, \theta) r \, dr \, d\theta}{\int \int \xi(r, \theta) r \, dr \, d\theta}, \text{ with } \xi \in [1, \rho u_{\text{ax}}, ...]. \tag{3.16}
\]

This is not an unambiguous process but \( \Phi(r, \theta) \) can be locally weighted with various quantities \( \xi \). As described by Cumpsty and Horlock [39] the appropriate weights are determined by the underlying problem in order to derive meaningful conclusions. However, in practical applications mostly either area \((\xi = 1)\) or mass flow \((\xi = \rho u_{\text{ax}})\) averaging is employed where mass flow averaging is to be preferred for most flow quantities.

---

\(^{10}\) The in-plane flow components \( u_{\text{rad}} \) and \( u_{\text{tan}} \) play a minor role because the stator is mostly affected by the incidence angle which is explicitly prescribed by the inlet BC.

\(^{11}\) The domain of the coupled simulation (by Hilgert et al. [73]) is similar to the domain shown in Fig. D.5, the domain of the downstream simulation is shown in Fig. 3.4.
0.6% $\overline{T_{t,\text{in}}}$ which corresponds to a difference in isentropic HPT efficiency of 0.2%. The problem can therefore not be avoided by falling back to area averaging.

The difference in mass flow weighted quantities is especially relevant if the same inlet profile is clocked to different circumferential positions. Then, the potential field of the vanes will have an upstream effect on inlet velocity that is geometrically fixed whereas the hot spots in the traverse will move with the clocked profile. More mass will therefore pass through hot spots clocked to the passage which increases their weight in the averaging and thus leads to a rise of $\overline{T_{t,\text{in}}}$ (Fig. 3.3 b).

### 3.3.3 Scaling Turbine Inlet Conditions

It is common practice to scale the turbine inlet temperature\textsuperscript{12} to design conditions in order to reach a thermodynamically comparable state for an assessment of turbine efficiency $\eta_{\text{HPT}}$. That is, the temperature at turbine inlet is adjusted so that the mean inlet temperature reaches a target level. In general, an increase in the averaged turbine inlet temperature has direct and indirect effects on $\eta_{\text{HPT}}$. On the one hand, it directly causes a decrease in $\eta_{\text{HPT}}$ which is not caused by an increase of entropy but which is an inherent consequence of the definition of isentropic efficiency (cf. Section 2.2.2). On the other hand, it may induce viscous losses from flow acceleration and higher loading of the rotor blades (cf. Fig. 2.7) which indirectly reduce $\eta_{\text{HPT}}$. The goal in the aerodynamic design of a HPT is to minimise these losses.

If turbine inlet temperature was not kept constant when analysing different designs, the indirect effects of these losses on turbine efficiency would not be distinguishable from the direct effects and one might misinterpret deviations of the operating conditions as increase or reduction of losses in the flow.

The turbine inlet temperature is typically evaluated on a plane in front of the NGVs (plane 40) which is defined as inlet for the control volume for efficiency definition. In order to maintain the turbine operating conditions, the temperature at CFD inlet is then, uniformly or locally, enhanced or reduced in an iterative process to set the mass flow averaged temperature

$$T_{t,40} := \frac{1}{m} \int \int_{40} \rho u_{ax} T_{t} r \, dr \, d\theta$$

in plane 40 to a predefined value. In this work, plane 40 coincides with the inlet of the CFD domain in order to simplify this process but this is not necessarily the case. $T_{t,40}$ is the most reasonable quantity to preserve in scaling from the point of view of the thermodynamic process in the turbine. From a systemic perspective, however, the conservation of enthalpy inflow $\dot{H}_{t,40}$ might be more appropriate which is discussed in Appendix B.3.1.

\textsuperscript{12} Commonly, only total temperature is scaled due to its large variation across the traverse at the order of 50% of its mean value as opposed to about 1% variation in total pressure.
3.4 Evaluation of the Flow Solver

All CFD results presented in this work are obtained using the commercial flow solver ANSYS® CFX® [29]. In order to evaluate the capabilities of the solver to adequately resolve the investigated flow phenomena, aero-thermal CFD results are compared to flow measurements in this chapter.

3.4.1 Test Case, Mesh and Numerical Set-up

The Large Scale Turbine Rig serves as test case for an evaluation of the flow solver. The LSTR is an isothermal, closed-loop, low-speed turbine test rig operated at TU Darmstadt to investigate effects of swirling combustor exit flow on aerodynamics, cooling and heat transfer in a 1.5 stage turbine. The swirling flow is generated in a swirl generator upstream of the turbine stage (Fig. 3.4).

![Diagram of LSTR Measurement Section and Numerical Model of NGV1]

**Figure 3.4:** The domain used for numerical simulations shown schematically in a cut through the measurement section of the test rig (top left) and as a 3D model, together with the distribution of \( y^+ \) (top right). The axial positions, where thermal data on the hub are compared to experiments, are highlighted as black dashed lines.

The turbine geometry is representative of real HPT hardware with 24 vanes in the NGV1 row, 36 squealer-tip rotor blades and 34 vanes in NGV2. It is scaled up by a factor of 3:1 to enable flow field measurements with high resolution. The scaling obeys a similarity of Reynolds number, Eqn. (3.2), with respect to realistic operating conditions (Re \( \approx 8 \times 10^5 \), based on NGV chord and exit velocity). Flow
Mach numbers are much lower than in a real engine (Ma ≈ 0.3 at rotor inlet) and the flow throughout the rig can be considered incompressible. Also, the flow in the combustor module is iso-thermal. The missing reaction is expected to cause different swirling modes than in reactive combustors (cf. Sec. 2.1.2). Further information on the rig’s design can be found in Krichbaum et al. [102].

The LSTR allows for investigations of the effects of operating conditions, Eqn. 2.18, on film cooling because coolant for the RIDN upstream of the vanes as well as the vane film and trailing edge cooling is provided and controlled independently with a difference of up to 20 K relative to the main flow. The RIDN injection consists of two rows of 19 holes (upstream) and 20 holes (downstream), per swirler, inclined 60° to the axial direction and 20° to the hub wall. Mass flow \( \dot{m} \) through the main annulus is at about 9.5 kg/s and an additional 0.9 kg/s of secondary air are provided. 4.9% of \( \dot{m} \) at NGV inlet (measurement plane ME01) enter the annulus as airfoil film cooling and 2.0% are ejected through seven discrete slots at the TE. Additional purge flow is ejected through the rotor wheel cavities. During operation, the reduced mass flow at ME01 and the reduced spool speed are maintained. The density ratio of coolant to main flow is not matched to engine conditions but the blowing rate BR is. The momentum ratio is therefore higher than in a real engine.

Swirl is generated by twelve swirlers containing three concentric rings of vanes which are modelled based on the E3E swirler. The nominal swirl number is \( S = 0.6 \).

The turbine inlet swirl angle is mitigated by a tapered liner to compensate for flow acceleration from combustion. The mean swirl angle is larger in the upper half of the annulus, as shown in Fig. 1.5. The swirlers in the combustor simulator are mounted to “clockable” rings which enables the generation of a swirling inflow to the turbine at different circumferential positions. Three configurations have been investigated experimentally: Axial inflow (Ax, no swirlers installed), swirling inflow with the combustor centre line geometrically aligned with LE of a vane at 50% span (swirl to leading edge, SwL) and swirling inflow aligned in between the LE of two adjacent vanes (swirl to passage, SwP).

The different measurement planes ME00 to ME05 for the acquisition of aerodynamic data are shown in Fig. 3.4. 2D flow field measurements by means of pneumatic and hot wire probes have been conducted in the planes ME01 to ME03 (ME01 corresponds to the CTI interface) in different measurement campaigns. In this work, the flow between ME01 and ME02 is considered for validation because detailed measurements of \( \eta_{FC} \) and wall HTC at the hub are available for this section only. These were measured by Werschnik [180] by means of infrared thermography using an auxiliary hub wall that was heated on one side to generate an approximately 1D heat flux through the hub surface. He also measured the flow field in planes, RE1 and RE2, in the proximity of the RIDN injection holes (Fig. 3.10), as well as vane surface static pressure distribution. Mean flow aerodynamics in ME01 and

3.4 Evaluation of the Flow Solver
ME02 from five hole probe measurements are reported by Hilgert et al. [73]. Hot wire measurements\textsuperscript{13} yielding the distribution of turbulence were conducted by Wilhelm et al. [186] by means of hot wire anemometry using split fiber probes. All results are compared at a RIDN mass flow ratio of $MFR = 3$. Since the rig is iso-thermal, approximately at ambient conditions, the RIDN coolant air was heated in the experiments (reversing the heat flux of the actual, cooled engine) in order to generate a driving temperature difference allowing for heat transfer measurements. The numerical set-up is based on the work of Ivanov [84]. The numerical domain is shown in Fig. 3.4. As the rig has 24 first-row vanes and 12 swirlers, a 30° segment of NGV1 can be modelled assuming periodic conditions at the boundaries. The main inlet is located at measurement plane ME01, the domain outlet is at ME03. The rotor blade is not modelled and the physical space downstream of ME02 is not considered in the comparison but only modelled to prevent a direct impingement of stator wakes on the outlet section. Numerical settings and mesh information can be seen in Tab. 3.1.

The low-Re unstructured mesh is generated using the software CENTAUR 10.6 [171] with 15 prismatic layers at the walls to resolve flow boundary layers and tetrahedra and pyramids in the rest of the domain. It is refined around the RIDN jets as well as in the wakes regions behind the vanes.

<table>
<thead>
<tr>
<th>Table 3.1: Numerical Set-up for Simulations of the Large Scale Turbine Rig</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Numerical Mesh</strong></td>
</tr>
<tr>
<td>Domain</td>
</tr>
<tr>
<td>Passages</td>
</tr>
<tr>
<td>Cells (Mio.)</td>
</tr>
<tr>
<td>Max. $y^+$</td>
</tr>
<tr>
<td>Avg. $y^+$</td>
</tr>
<tr>
<td>Max. EVR\textsuperscript{⋆}</td>
</tr>
<tr>
<td>Avg. EVR</td>
</tr>
<tr>
<td>Min. cell angles (°)</td>
</tr>
</tbody>
</table>

\textsuperscript{13} These measurements were conducted at a different rig operating point than the previous measurements. Since the qualitative flow pattern in ME01 and ME02 is similar to that of the previous measurements, the offset in operating conditions is ignored in the CFD comparison of turbulence in this work.
3.4.2 Axial Inflow

Aerodynamics and Vane Loading

CFD results are first compared with experiments obtained with the axial inflow condition. A comparison of circumferentially averaged data at measurement planes ME01 and ME02 is shown in Fig. 3.5 at different resolutions of the numerical grid, in order to illustrate the effect of the discretisation error. Measured conditions in ME01 are used as inlet conditions for CFD (except for $\alpha_{\text{tan}}$ which was set to zero). The data in ME02 can thus be used to assess the performance of the solver.

![Graph showing comparison of predicted, circumferentially averaged flow quantities on different meshes (between 27.5 and 50.8 Mio cells) with experiments in ME02 at axial inflow. The RSM-BSL turbulence model is used.](image)

Figure 3.5: Comparison of predicted, circumferentially averaged flow quantities on different meshes (between 27.5 and 50.8 Mio cells) with experiments in ME02 at axial inflow. The RSM-BSL turbulence model is used.

It can be seen, that the data in ME02 are qualitatively matched well by the numerical simulation. The influence of mesh resolution on aerodynamic data is small (in comparison to thermal data shown below). Quantitatively, deficits between CFD and experiment at ME02 in total pressure at 15% and 40% span are apparent. These correspond to radial positions where the main flow interacts with the rim seal injection and where RIDN flow, washed-up at the vane PS, exits the passage (Werschnik et al. [183]), respectively. Moreover, a large quantitative offset in the prediction of turbulence intensity is visible. It is assumed that this discrepancy to experimental values is mainly due to the missing blade row interaction between stator and rotor vanes which is not modelled in the CFD.
A comparison of blade loading in terms of *isentropic* Mach number\(^{14}\) at the vane surface for the finest mesh (50.8 Mio. cells) is shown in Fig. 3.6. The data agree well, except for a region at the vane SS at \(c_{\text{ch}} \approx 0.5\). Similar discrepancies have been observed in literature (e.g. Griffini et al. [58]) and might be attributed to unsteady vortex shedding which cannot be reproduced correctly by steady simulations.

**Figure 3.6:** Blade loading of NGV1 at three span heights. The pressure side curve is cut-off at the trailing edge cutback at 50% and 20% span for the purpose of clear visualisation. Locally large values of \(M_{\text{is}}\) are due to film cooling ejection.

**Film Cooling Effectiveness**

The findings on the flow field in the NGV1 passage by Werschnik et al. [183, 181, 182] are summarised first to help interpreting the results. A qualitative comparison of \(\text{Nu}\) and \(\eta_{\text{FC}}\) on the hub between CFD and experiment at axial inflow is shown in Fig. 3.7. The flow from the first (upstream) RIDN row is detached from the wall, reducing the effective blowing rate of the second row, which therefore stays attached to the wall. A streak pattern is visible in the entrance region to the passage where coolant is sucked away from the wall in-between two adjacent holes by the induced kidney vortices of the second row. A large part of RIDN coolant is carried, by its large momentum, across the passage to the PS/hub corner where it accumulates and causes a high film cooling effectiveness. Hub wall \(\text{Nu}\) is increased at the suction side shoulder, where the horse-shoe legs merge, and in the region of highest velocity in the throat cross section. The axial point of transition between low and high \(\text{Nu}\) was found to change with RIDN MFR and inlet swirl by Werschnik [180].

Film cooling effectiveness \(\eta_{\text{FC}}\) at the hub is shown in Fig. 3.8 at different axial planes (locations are illustrated in Fig. 3.4) for four numerical grids with different resolution. The three meshes with 27.5 to 45.8 Mio cells, referenced in Fig. 3.8, are refined approximately equally within the flow domain and at the bounding walls.

---

\(^{14}\) A hypothetical Mach number at the vane surface assuming isentropic flow and slip walls. Measurements are conducted by Werschnik [180].
Figure 3.7: Comparison of measured and predicted distributions of film cooling effectiveness $\eta_{FC}$ (left) and Nusselt number $\text{Nu}$ (right) at the hub end wall for axial inflow.

Figure 3.8: Comparison of predicted film cooling effectiveness $\eta_{FC}$ on different meshes (between 27.5 and 50.8 Mio cells) with experiments at several axial cuts reported as distance $\Delta x$ to the trailing edge of the last film cooling row normalised by hole diameter $D$. The RSM-BSL turbulence model is used with $Pr_t = 0.15$.

The final mesh with 50.8 Mio cells was refined (with reference to 45.8 Mio cells) exclusively on the hub end wall which can be seen to greatly improve the predictions qualitatively and quantitatively. A further refinement of the hub wall led to failure of the mesh generation process which is why this mesh was chosen as final mesh for further simulations. A refinement of the hub boundary layer (transverse to the wall) did not significantly alter the results.
The uncertainty of the experimental data is estimated to be approximately 8\% at an effectiveness level of $\eta_{FC} = 30\%$ by Werschnik [180]. The predictions are outside this uncertainty band in the first plane at $\Delta x = 3D$ for all mesh resolutions. The prediction of $\eta_{FC}$ in this region is highly dependent on the modelling of turbulent thermal diffusion transverse to the film cooling jet which is assumed to be the main reason for deviations between experiment and CFD. In the turbulence models used, turbulent thermal diffusion is modelled using a gradient diffusion approach, Eqn. 3.7, with a thermal diffusivity inversely proportional to the turbulent Prandtl number $Pr_t$. An effect of this parameter on predicted film cooling efficiency is reported by Jones et al. [92].

As shown in Fig. 3.9, the influence of $Pr_t$ is especially large in the proximity of the coolant jet ejection (at $\Delta x/D = 3.0$). Best agreement with experimental data is achieved for $Pr_t = 0.15$ (as opposed to the solver’s default value 0.9). A variable formulation $Pr_t = f(Re_y)$, proposed by Jones et al. [92], does not yield an improvement of the predictions of $\eta_{FC}$ with the RSM-BSL model. It can be seen, that the influence of $Pr_t$ is much larger than that of the turbulence model. Also, the effect of $Pr_t$ on film cooling effectiveness at the hub has shown to overshadow other effects (compressibility, inlet turbulence intensity and length scale, inlet boundary layer thickness and the number of cells in the boundary layer).

In order to evaluate the influence of the momentum of film cooling jets on the prediction of $\eta_{FC}$, the local variation in Mach number ratio $MR$ across the jets in circumferential direction, measured by Werschnik [180], is compared with CFD in Fig. 3.10. The qualitative influence of the potential field of the vanes is in good agreement with the measurements. Quantitatively, the extrema are offset by 5\% pitch and an offset in the level of $\Delta MR \approx 0.5$ is recognised. It is assumed that this difference is mainly due to a systematic error in the prediction of aerodynamic blockage within the film cooling holes that increases the jet momentum in the simulations (jetting effect described by Leylek and Zerkle [112]).
Figure 3.10: Definition of evaluation planes RE1 and RE2 of the local Mach number ratio MR across the RIDN cooling holes (left, centre) and comparison of MR with experiments (right)

Heat Transfer

In addition to the adiabatic effectiveness of film cooling, heat transfer at the end walls in terms of Nu distribution is compared with experiments. Results are shown in Fig. 3.11.

Figure 3.11: Comparison of predicted Nusselt number Nu at several axial cuts for different mesh resolutions and turbulence models

As for the film cooling effectiveness, a strong dependence on turbulent Prandtl number is recognised. It is set to its default value of \( Pr_t = 0.90 \) for these simulations. With \( Pr_t = 0.15 \) differences to the measurements are much larger (at the order of

3.4 Evaluation of the Flow Solver 61
100 % of Nu). It can be seen that heat transfer is strongly underpredicted by CFD at axial positions up to $\Delta x = 12.9D$ with maximal differences in Nu of approximately 50 % but simulations match experimental values better in the rear part of the passage. The uncertainty of the experimental data is estimated to be approximately 9.5 % at a level of $\text{Nu} = 1900$ by Werschnik [180].

3.4.3 Swirling Inflow

Aerodynamics

The solver evaluation proceeds with comparisons at swirling inflow conditions, clocked to the vane LE (SwL). The 2D inlet conditions for CFD are not taken from experiments, as in the previous section, but from the coupled SAS by Hilgert et al. [73] which provides a consistent and perfectly periodic data set at ME01. As can be seen in Fig. 3.12, these numerically obtained inlet conditions are in fairly good agreement with the measurements.

Figure 3.12: Comparison of predicted, circumferentially averaged flow quantities with experiments in ME01 and ME02 at swirling inflow (SwL) on the 50.8 Mio cell mesh. Inlet conditions at ME01 are retrieved from the simulation by Hilgert et al. [73].

The flow conditions at ME02, predicted by different turbulence models, qualitatively represent all relevant features of NGV exit flow. Largest deviations in $p_t$ occur between 30 and 70 % span. These might be due to effects of inlet swirl (similar differences can be seen in the LES by Jacobi et al. [88]) and an insufficient prediction of the flow exiting the vane through the TE slots which may be deteriorated by the quality of the numerical mesh in the boundary layers of the slot, or the prediction of the internal cavity flow exiting the vane. Deviations in exit flow angle $\alpha_{\tan}$ above 70 % span may be ascribed to differences in the inlet condition. The performance of
the SST-$k$-$\omega$ model is similar to the two investigated RSMs. The EARS model performed far worse than the other models. A more detailed comparison of the 2D flow features in ME02 can be found in Section 5.3.1.

Blade loading, in terms of isentropic Mach number $Ma_{is}$, is compared in Fig. 3.13 for MFR = 0. The data are in good agreement at all sections where measurement data are available. The characteristic aerodynamic loading of the left vane that is directly exposed to swirl is recognisable (cf. Section 5.2.3). Positive incidence loads the vane at 80 % span and negative incidence unloads it at 20 % span.

![Figure 3.13: Blade loading of NGV1 at three span heights for SwL and no coolant injection on the 50.8 Mio cell mesh. Locally large values of $Ma_{is}$ are due to film cooling ejection.](image)

**Film Cooling Effectiveness**

Distributions of film cooling effectiveness $\eta_{FC}$ are shown in Fig. 3.15 for five different combinations of turbulence model and $Pr_t$. Due to the inhomogeneity of the inlet conditions, two neighbouring passages are shown. Inlet swirl in counter-clockwise direction is clocked to the vane left of the left passage ($\theta \approx -12^\circ$ at $\Delta x = 0$), i.e., the left passage experiences an upwash and the flow in the right passage is washed down.

The overall agreement between CFD and experiment is worse than for axial inflow boundary conditions with differences $\Delta \eta_{FC}$ between 10 and 20 %. In general, CFD
under-predicts the mixing of the cooling jets and the footprints of individual cooling
holes can be identified in CFD over much greater axial distance (up to $\Delta x = 19.6D$)
than in experiments ($\Delta x = 12.9D$). This effect is most clearly visible in the entrance
region of the left passage (from $\Delta x = 6.3D$ to $12.9D$) where the flow is influenced
by the upwash from inlet swirl. There, a higher maximal $\eta_{FC}$ than in the right
passage is falsely predicted. Since the influence of $Pr_t$ is rather small in this region,
the effect is probably attributed to unsteady fluctuations which are not resolved
by the steady inflow conditions. An increased film cooling effectiveness in the
hub/PS corners in the rear part of the passage (cf. Fig. 3.7) can be observed both in
experiments and CFD (for most turbulence models).
As in the case of axial inlet conditions, $\eta_{FC}$ depends strongly on $Pr_t$. The front part
of the left passage is matched better if $Pr_t = 0.9$, the rear part is matched better if
$Pr_t = 0.15$ and vice versa in the right passage. The RSM-BSL shows the best overall
agreement at almost all investigated positions.
The local blowing rates MR, measured by Wer-
schnik [180] for swirling inflow, are compared
to both, the decoupled turbine simulations with
the domain shown in Fig. 3.4 as well as the
coupled simulation by Hilgert et al. [73], in Fig.
3.14. The blowing rates, locally elevated by
inlet swirl in the experiments, are reproduced
by neither of the simulations. The difference is
assumed to result mainly from modelling errors
in CFD. As both, the coupled simulation with
an unsteady inlet traverse and the steady simulation with a steady inlet traverse, do
not show the local elevation of MR, a possible explanation might be sector to sector
interaction of different swirlers which is suppressed by assuming a periodic BC in
CFD. This effect is outside the scope of this investigation and effects associated with
this mismatch must be regarded as modelling error to account for in the subsequent
uncertainty analyses.

Heat Transfer
A comparison of Nu for swirling inflow is shown in Fig. 3.16. CFD was conducted at
$Pr_t = 0.9$. The qualitative agreement in Nu is better than that of $\eta_{FC}$. This indicates
that mismatches in $\eta_{FC}$ are caused rather by the modelling of turbulent diffusion
at the wall than a false prediction of the flow field. Again, it can be recognised
that discrete coolant streaks are conserved over larger axial distance than in the
experiments and the largest differences in Nu occur in the proximity of the cooling
holes.
Figure 3.15: Comparison of film cooling effectiveness $\eta_{FC}$ at several axial cuts for different turbulence models and values of $Pr_t$ on the 50.8 Mio. cell mesh. Only the RSM-BSL results are shown in the first two plots for the purpose of clear visualisation.
Figure 3.16: Comparison of Nusselt number $\text{Nu}$ with swirling inflow at several axial cuts on the 50.8 Mio. cell mesh with $Pr_t = 0.9$
3.4.4 Summary of Solver Evaluation and Impact on further Studies

Aerodynamic effects in the turbine are reproduced by the flow solver with mostly satisfactory results for axial as well as for swirling inflow conditions. Film cooling efficiency $\eta_{FC}$ and Nusselt number $Nu$ at the hub show a strong dependence on mesh resolution at the wall and the modelling of turbulent diffusion, i.e., the value of the turbulent Prandtl number $Pr_t$. Experimental data of the configuration with axial inflow can be matched throughout the passage with constant values for $Pr_t$. However, different constants are required for matching $\eta_{FC}$ ($Pr_t = 0.15$) and $Nu$ ($Pr_t = 0.9$).

With swirling inflow, $\eta_{FC}$ data are located in between the predictions with these constant values for $Pr_t$. Therefore, an improved modelling of the local turbulent diffusion is needed to better match results with swirling inflow. For the realistic cases with hot streaks simulated in the next chapters, the solver value of $Pr_t$ is not adjusted in order to prevent an unrealistic diffusion of hot streaks away from the wall.

The observed differences, at the order of 10 to 20% in $\eta_{FC}$, are regarded as an epistemic uncertainty of the turbulence model. In subsequent analyses, uncertainties of adiabatic wall temperature due to changes in inlet conditions can therefore only be regarded significant if they are larger than the equivalent of $\Delta \eta_{FC} = 20\%$. 
3.5 Uncertainty Quantification Methods

3.5.1 Review of Fundamental Statistical Properties and Definitions

The simple “experiment” of rolling a dice is used to illustrate fundamental statistical properties in this section which are reproduced from Dekking [42]. Due to different kinds of imperfections, it is practically impossible to accurately predict the number of pips that will result from rolling a dice. Hence, it can be considered a random experiment with uncertain outcome. Being a random experiment, it has a sample space $\Omega$ - a set whose elements $\omega$ describe the possible outcomes of the experiment. In the considered example, the sample space covers all possible pips of a dice throw, i.e., $\Omega = [1, 2, 3, 4, 5, 6]$.

From this sample space events can be derived. An event $A$ is a subset of $\Omega$. In the example, a number of different events can be defined such as event $A_1$, “the number thrown is larger than two.” From experience we know that if a fair dice is thrown sufficiently often and the results are recorded, eventually, the portion of each pip will approach $1/6$ of all throws, i.e., each number of pips is equally probable. From that, the probability of the event $A_1$ can be computed as $P(A_1) = \frac{4}{6} \approx 66.67\%$. Similarly, we can compute the probabilities of other events $A_2$, “the number thrown is smaller than three”, $A_3$, “the number thrown is either one or five” and so on. A function $P$ that assigns all events $A$ on the sample space $\Omega$ a probability $P(A)$ in $[0, 1]$ is called probability function. Two events $A$ and $B$ are called independent if the conditional probability of $A$ given $B$ is equal to the probability of $A$.

In the experiment considered above, the sample space $\Omega$ contains numeric elements. In the general case elements could also be non-numeric data. An example is the experiment of randomly picking a month from a calendar with the sample space $\Omega = \{\text{January}, \text{February}, \ldots, \text{December}\}$. The elements $\omega$ of this sample space $\Omega$ are transformed to real numbers in $[1, 2, \ldots, 12]$ by a random variable $X : \Omega \rightarrow \mathbb{R}$. Hence, the probability of $X$ assuming a value $x \in \mathbb{R}$ corresponds to the probability that $\omega \in \Omega$ occurs. Similarly, each event $A$ is mapped to an interval in $\mathbb{R}$.

If the outcomes of an experiment are discrete numbers, such as in the dice rolling experiment, $X$ is a discrete random variable. If they may be represented on a continuous scale, $X$ is called a continuous random variable. Most results of physical experiments, such as a flow velocity $u$ measured by a probe, can be regarded as continuous random variables. If $X$ is continuous, the function $f(x)$ defined by

$$P(a \leq X < b) = \int_a^b f(x) \, dx,$$

(3.18)

with $a \leq b$, $f(x) \geq 0 \forall x$ and $\int_{-\infty}^{+\infty} f(x) \, dx = 1$.
is called \emph{probability density function} (PDF). It can be seen from Eqn. (3.18) that the area under $f(x)$ between two points $a$ and $b$ corresponds to the probability of $X$ lying between these two values. The integral $F(x) = \int_{-\infty}^{x} f(t) \, dt$ of the PDF is referred to as \emph{cumulative density function} (CDF). One of the best known and physically most important\footnote{According to the central limit theorem the sum of a large number of random variables with arbitrary distribution approaches a normal distribution which explains its importance in the modelling of physical processes.} PDFs is that of a normally distributed variable

$$f(x) = \frac{1}{\sigma \sqrt{2\pi}} \exp \left[ -\frac{1}{2} \left( \frac{x-\mu}{\sigma} \right)^2 \right] \text{ for } -\infty < x < \infty$$

with the parameters $\sigma^2$ and $\mu$ which represent the distribution’s variance and expectation, as defined in Eqn. (3.20). Other well-known distributions can be found in the literature, e.g. Voigt [177].

From an engineering perspective, one is rarely interested in the detailed distribution $f(x)$ of an uncertain physical quantity but rather in what the average value of that quantity is and how much of a deviation from that value can be expected. These properties are quantified by the expectation $E[X]$ and variance $\text{Var}(X)$ of a random variable $X$. For continuous random variables these are defined as

$$E[X] := \int_{-\infty}^{\infty} x f(x) \, dx \quad \text{and} \quad \text{Var}(X) := E[(X - E[X])^2],$$

respectively. In practice, often the standard deviation $\sigma = \sqrt{\text{Var}(X)}$ is used instead of the variance as it has the same dimension as the expectation $E[X]$.

The \emph{correlation coefficient} $\rho$, according to Pearson, is defined as the covariance of two random variables $\text{Cov}(X,Y)$ normalised by their respective standard deviations

$$\rho(X,Y) := \frac{\text{Cov}(X,Y)}{\sqrt{\text{Var}(X)} \sqrt{\text{Var}(Y)}} = \frac{1}{N-1} \sum_{k=1}^{N} (X - E[X])(Y - E[Y]) \frac{\sqrt{\text{Var}(X)} \sqrt{\text{Var}(Y)}}{\sqrt{\text{Var}(X)} \sqrt{\text{Var}(Y)}}$$

and serves as a measure for statistic relation between two random variables. It varies in a range $[-1, 1]$. That is, $|\rho| = 1$ implies a perfect linear (positive or negative) \emph{correlation}\footnote{It is important to note that $\rho$ can only detect \emph{correlation}, not \emph{causality}. A large value of $\rho$ does \emph{not} imply that a change of $X$ is the \emph{cause} for an effect in $Y$ or vice versa because both variables could be affected by a third variable.} of $X$ and $Y$, whereas small correlations of, say, $|\rho| < 0.3$ indicate uncorrelated data. Examples of correlated and uncorrelated data sets are shown in Fig. 3.17 (right).

\emph{Significant} correlations (with a small probability of falsely rejecting the \emph{null hypothesis}, the $p$-value, below 0.05) are indicated by the superscript “**” in the following.

3.5 Uncertainty Quantification Methods
3.5.2 Classification of Uncertainty in Technical Systems

In the description of technical systems uncertainties are distinguished from errors. According to Oberkampf et al. [129], an error is “a recognisable deficiency in any phase or activity of modelling and simulation that is not due to lack of knowledge” whereas an uncertainty “(...) is due to lack of knowledge”. With regard to CFD, errors are subsumed in the numerical error introduced in Section 3.1.5. According to Montomoli et al. [123] uncertainties are further divided into aleatory uncertainties which account for an irreducible variability of physical properties (material properties, tolerances and boundary conditions\textsuperscript{17}) and epistemic uncertainties which account for a lack of knowledge of the physical modelling (turbulence models, steady state and periodicity assumptions, etc.). Aleatory uncertainties can be reduced by means of uncertainty quantification methods whereas epistemic uncertainties can only be reduced by continuous improvement of the underlying models.

3.5.3 Uncertainty Quantification

In the context of UQ, a system $\eta$ is analysed connecting input quantities $x_1, x_2, \ldots, x_n$, that are collected in the vector $x$, with output quantities $y_1, y_2, \ldots, y_m$, collected in $y$,

$$y = \eta(x).$$ \hspace{1cm} (3.22)

The system $\eta$ represents an arbitrary set of equations, modelling a physical problem. In the context of this work, $\eta$ models the turbulent flow through a HPT (cf. Section 3.1.2), $x$ determines the turbine inlet boundary condition and $y$ represents the HPT’s performance.

Under idealized conditions, without any uncertainty attributed to the parameters,

\textsuperscript{17} Reasons for aleatory uncertainties of HPT inlet conditions are discussed in Section 1.1.5.
there is a single, definitive input vector $\mathbf{x}$ and output vector $\mathbf{y}$. As described in Section 1.1.5, however, the input $\mathbf{x}$ deviates from its nominal value with a certain probability $P$ which can be described by a PDF (Eqn. (3.18)). The elements of $\mathbf{x}$ must therefore be regarded as random variables $X_1, X_2, \ldots, X_n$ collected in $\mathbf{X}$. The uncertainties in $\mathbf{X}$ are transported through the system $\eta$, leading to uncertainties of the outputs $\mathbf{Y}$ and transforming the deterministic system (3.22) into a probabilistic one

$$\mathbf{Y} = \eta(\mathbf{X}).$$

(3.23)

The goal of uncertainty quantification (UQ) methods is providing the statistics of $\mathbf{Y}$ from a minimal number $N$ of evaluations of the probabilistic system (3.23) given the uncertain input $\mathbf{X}$.

Since, in the context of this work, each evaluation of Eqn. (3.23) requires the expensive execution of a 3D CFD simulation, the applied UQ methods are supposed to be as efficient as possible. In the following paragraphs, two methods of uncertainty quantification, Monte Carlo sampling and polynomial chaos expansions, are introduced. The formulations described cover non-intrusive variants only which can be coupled with “black box”, commercial CFD solvers. The effect of epistemic uncertainties inherent to the solver (Section 3.5.2) must, however, be considered when interpreting the data.

### 3.5.4 Monte Carlo Sampling

The Monte Carlo method\(^{19}\) was first used for the modelling of the diffusion of neutrons by the group around Stanislav Ulam and John von Neumann that was instructed with the development of the first atomic bomb in the Manhattan Project. Today, the technique covers a large range of applications in the fields of mathematics, economics, science and engineering. The most widespread application of Monte Carlo Sampling is the efficient solution of highly-dimensional integrals. In the following, the aspects of Monte Carlo Sampling (MCS) in the context of UQ will be discussed.

The basic idea of MCS is to obtain an approximation $\mathbf{Y}$ of the output vector $\mathbf{Y}$ in Eqn. (3.23) by creating $N$ discrete samples of $\mathbf{X}$ and evaluating the system $\eta$ for each sample. That is, $N$ sampling points are generated for each of the $n$ elements $X_1, X_2, \ldots, X_n$ of $\mathbf{X}$, such that the subset of $N$ discrete samples has the same statistical properties as the whole population yielding the combinations $x_1, x_2, \ldots, x_N$. The system is then evaluated for each combination and, from the outputs $y_1, y_2, \ldots, y_N$, the

\(^{18}\) In order to distinguish random variables $X$ from deterministic variables $x$ they are represented by capital letters.

\(^{19}\) The name Monte Carlo was originally suggested by Metropolis [119] as a reference to the gambling tradition in the city Monte Carlo.
the statistical properties of $\tilde{Y}$ are inferred. By the Law of large numbers a sufficiently large number of evaluations $N$ will eventually yield the exact distribution of $Y$ and

$$\lim_{N \to \infty} E[\tilde{Y}] = E[Y] \quad \text{and} \quad \lim_{N \to \infty} \text{Var} (\tilde{Y}) = \text{Var}(Y). \quad (3.24)$$

The method is illustrated by computing the propagation of uncertainty of high pressure turbine inlet temperature $T_{t,40}$ and efficiency $\eta_{\text{HPT}}$,

$$X = [T_{t,40}, \eta_{\text{HPT}}], \quad (3.25)$$

through the thermodynamic cycle of a turbofan engine (cf. Fig. 1.1). Computations of the cycle (the probabilistic system $\eta$ in Eqn. (3.23)) are carried out using the commercial tool G\textit {AS}TURB 12 [172]. Since no complete data sets of real commercial jet engines are available in open literature, the cycle to be analysed is taken from a representative text book example (example 4.13 in Farokhi [48], properties are given in Appendix B.6.1). The uncertain inputs $X$ are assumed to be normally distributed with $\sigma(T_{t,40}) = 1.0\%$ and $\sigma(\eta_{\text{HPT}}) = 0.2\%$.

The histograms in Fig. 3.19 illustrate the distribution of sample points of $X_1 \equiv T_{t,40}$ and $X_2 \equiv \eta_{\text{HPT}}$ for $N = 10, \ldots , 10\,000$ cycle analyses. The propagation is evaluated with respect to engine’s thermal and propulsive efficiency, $\eta_{\text{th}}$ and $\eta_{\text{pr}}$, and specific core net thrust $F_{\text{n,core}}/\dot{m}_\infty a_\infty$.

$$Y = \left[ \eta_{\text{th}}, \eta_{\text{pr}}, \frac{F_{\text{n,core}}}{\dot{m}_\infty a_\infty} \right]. \quad (3.26)$$

The statistical moments $E[\tilde{Y}]$ and $\text{Var}(\tilde{Y})$ are converged at $N = 1000$. Regarding $\eta_{\text{th}}$, it can also be seen that, due to non-linearity of the system, the outputs $Y$ are not necessarily distributed symmetrically even if all inputs $X$ are, but they may be skewed. This implies, that expected values of uncertain outputs $E[Y]$ are not necessarily equal to the datum output $y$ in Eqn. (3.22).

\footnote{Note, that thrust per fuel mass flow $F_{\text{n,core}}/\dot{m}_f$ is proportional to overall engine efficiency $\eta_o$ which is the product of $\eta_{\text{th}}$ and $\eta_{\text{pr}}$.}
The fundamentals of the approach are outlined in the following. Comprehensive uncertainty quantification methods are required to reproduce the assumed distributions of the uncertain parameters. However, for problems with large dimensionality the number of required model evaluations \( N \) grows quickly which is referred to as the Curse of Dimensionality. For a given dimension \( n \), the number of required evaluations \( N \) can be reduced by the application of advanced sampling techniques. These are required to reproduce the assumed distributions of the uncertain parameters \( X \) and to prevent correlations among the samples of the different \( X \). Their individual correlation with the outputs \( Y \), according to Eqn. (3.21), would be indistinguishable otherwise. The sampling techniques used in this work, Hammersley (HS) and Latin Hypercube sampling (LHS), are described in Appendix B.4.

### 3.5.5 Generalised Polynomial Chaos Expansions

Generalised Polynomial Chaos Expansions (gPC) can be used as an alternative to MCS for retrieving statistical moments, Eqn. (3.20), of the uncertain outputs \( Y \) in Eqn. (3.23). The method is computationally more efficient than MCS, i.e., fewer model evaluations \( N \) are needed to obtain results with a given error for a small number \( (n \approx 5) \) of uncertain parameters \( X \) (Montomoli et al. [123]). The fundamentals of the approach are outlined in the following. Comprehensive

---

**Figure 3.19:** Monte Carlo simulation of engine cycle parameters with uncertain turbine inlet temperature \( \sigma(T_{i0}) = 1.0\% \) and HPT efficiency \( \sigma(\eta_{HPT}) = 0.2\% \) at different \( N \). Uncertain inputs \( X \) are sampled using Latin Hypercube sampling. The bar width illustrates the density of sampling points.

Often, one is interested in problems with higher dimensionality \( n \), i.e., more than \( n = 2 \) uncertain parameters. However, for problems with large dimensionality the number of required model evaluations \( N \) grows quickly which is referred to as the *Curse of Dimensionality*. For a given dimension \( n \), the number of required evaluations \( N \) can be reduced by the application of advanced sampling techniques. These are required to reproduce the assumed distributions of the uncertain parameters \( X \) and to prevent correlations among the samples of the different \( X \). Their individual correlation with the outputs \( Y \), according to Eqn. (3.21), would be indistinguishable otherwise. The sampling techniques used in this work, *Hammersley* (HS) and *Latin Hypercube* sampling (LHS), are described in Appendix B.4.

#### 3.5 Uncertainty Quantification Methods
reviews on the underlying theory can be found in O’Hagan [131], Najm [127] and Montomoli et al. [123]. The random variables in Eqn. (3.23) are approximated by \( \tilde{X} \) and \( \tilde{Y} \) such that

\[
\begin{align*}
\tilde{X} & \approx X \\
\tilde{Y} & \approx Y
\end{align*}
\]

\( \Rightarrow \quad \tilde{Y} \approx \eta(\tilde{X}) \quad \Rightarrow \quad \{ E[\tilde{Y}] \approx E[Y] \}
\]
\[\text{Var}(\tilde{Y}) \approx \text{Var}(Y). \tag{3.27}\]

The approximation \( \tilde{X} \) can be thought of similar to a truncated Fourier series: Random inputs \( X \) are represented by infinite series of polynomials \( \Psi_k(\xi) \) of order \( p \) with weights \( x_k \) that are truncated after \( P \) terms,

\[
X(\omega) = \sum_{k=0}^{\infty} x_k \Psi_k(\xi) \approx \tilde{X}(\omega) = \sum_{k=0}^{P} x_k \Psi_k(\xi), \quad \tag{3.28}
\]

where the basis \( \xi \) is a vector of independent random variables \( \xi(\omega) \) with the same dimension \( n \) and distribution as the uncertain input \( X \). The approximation \( \tilde{X}(\omega) \) in Eqn. (3.28) then converges exponentially towards \( X(\omega) \) (Najm [127]). The series in Eqn. (3.28) are referred to as Polynomial Chaos (PC) expansions based on the work of Wiener [184]. His approach was expanded to general, i.e. not necessarily normal, PDFs by Xiu [188], referred to as generalised PC. In the same way, an expansion \( \tilde{Y}(\omega) \) for outputs \( Y \) is constructed from the polynomials \( \Psi_k(\xi) \) from Eqn. (3.28) and unknown weights \( y_k \).

The efficiency of the method stems from the selection of the polynomials \( \Psi_k(\xi) \) which are constructed to be orthogonal polynomials, weighted with the PDF of the basis \( f(\xi) \), as defined by

\[
\langle \psi_i, \psi_j \rangle := \int \psi_i(\xi) \psi_j(\xi) f(\xi) d\xi = \delta_{ij} \langle \psi_i^2 \rangle. \tag{3.29}
\]

That is, the choice of polynomials \( \Psi_k(\xi) \) depends on the PDF \( f(\xi) \) of the uncertain input parameters. A list of common PDFs with their corresponding polynomials \( \Psi_k(\xi) \) can be found in Montomoli et al. [123]. It can be shown\(^{22}\) that, with this choice of the \( \Psi_k(\xi) \), the statistical moments of \( Y \) directly correspond with the coefficients \( y_k \) (Montomoli et al. [123]) and, in particular,

\[
E[\tilde{Y}] = y_0 \quad \text{and} \quad \text{Var}(\tilde{Y}) = \sum_{k=1}^{P} y_k^2 \langle \psi_k^2 \rangle. \tag{3.30}
\]

There are different approaches of retrieving the unknown weights \( y_k \), the best known of which are quadrature and collocation methods. The latter is used in this work.

\(_{21}\) The sign “\( \approx \)” in Eqn. (3.27) to (3.28) expresses an equality of the statistical properties of the random variables. Details can be found in O’Hagan [131].

\(_{22}\) When comparing Eqn. (3.29) with the moments (3.20) it becomes evident that the orthogonal inner product \( \langle \psi_i, \psi_j \rangle \) is, by definition, equal to the expectation \( E[\psi_i \psi_j] \).
due to its robustness with respect to the imperfections of CFD (numerical errors or missing samples due to convergence problems). The principle of quadrature methods is described in Appendix B.5.

Collocation Methods

Collocation methods use \( N \) samples \( \tilde{Y}(\xi_i) \) of the expansion of \( Y \) in order to set up a linear system of equations which, for a single element \( \tilde{Y} \) of \( \tilde{Y} \), reads

\[
\begin{bmatrix}
\Psi_0(\xi_0) & \Psi_1(\xi_0) & \cdots & \Psi_P(\xi_0) \\
\Psi_0(\xi_1) & \Psi_1(\xi_1) & \cdots & \Psi_P(\xi_1) \\
\vdots & \vdots & \ddots & \vdots \\
\Psi_0(\xi_N) & \Psi_1(\xi_N) & \cdots & \Psi_P(\xi_N)
\end{bmatrix}
\begin{bmatrix}
y_0 \\
y_1 \\
\vdots \\
y_N
\end{bmatrix}
= 
\begin{bmatrix}
\tilde{Y}(\xi_0) \\
\tilde{Y}(\xi_1) \\
\vdots \\
\tilde{Y}(\xi_N)
\end{bmatrix}
\tag{3.31}
\]

and can be directly solved for the \( y_k \). The \( N \) collocation nodes \( \xi_i \), required to set up the system, can be distributed using the sampling methods described in Appendix B.4. The minimal number of model evaluations, in order to obtain a closed system of equations, is

\[
P + 1 = \frac{(n + p)!}{n!p!}.
\tag{3.32}
\]

Hosder and Walters [75] recommend oversampling by a factor of \( n_P := \frac{N}{P+1} = 2 \) (that is, the number of evaluated collocation points \( N \) is twice as high as the minimal number \( P + 1 \), yielding an overdetermined system of equations) in combination with a regression scheme to solve Eqn. (3.31) in order to provide a more robust solution and compensate for outliers in the CFD. The process of transforming uncertain inputs \( X \) to outputs \( Y \) is summarised in Fig. 3.20.

**Figure 3.20:** Workflow of a UQ analysis using gPC with point collocation \((n = 1)\)

Implementation and Verification

The statistical pre- and post-processing of the gPC expansions is done using the **Python** library CHAOSPY by Feinberg and Langtangen [49]. In order to verify the
implementation of the collocation method, the UQ analysis of an engine cycle, shown in Fig. 3.19, is repeated with a second order \((p = 2)\) gPC analysis with \(n_p = 2\). Using gPC, the uncertain outputs converge to the MCS results (exact to the fourth digit) with \(N = 12\) evaluations, compared to \(N > 100\) with MCS. That is, for this low dimensional problem gPC is at least 10 times more efficient than a Monte Carlo analysis.

The effects of the gPC polynomial order \(p\) and the choice of regression scheme on the results have been analysed in a problem representative for the studies conducted in this work (uncertainty of film cooling effectiveness \(\eta_{FC}\) on HPT vanes in the LSTR) with \(n = 2\). The analysis is shown in Appendix B.7. A ridge regression scheme (Rifkin and Lippert [146]) showed to be more robust against overshooting in the solution of system (3.31) than an ordinary least squares regression. Also, a polynomial order of \(p = 2\) showed to be a good compromise between accuracy and stability of the solution. These settings are therefore employed in the analyses of the following chapters.
4 Method Development
Combustor Exit Flow Model

4.1 Introduction

4.1.1 Overall Modelling Approach

The goal of this work is to quantify the propagation of uncertainty of 2D inlet
traverses through HPTs by means of the UQ methods introduced in Section 3.5.
These probabilistic methods require \( N \) different inlet traverses. A suitable data base
of sufficiently many different combustor exit traverses is not available. Therefore, in
this work, these are produced by statistical sampling of a physical flow model with
a set of input parameters \( X \) (cf. Fig. 1.6).
The model must output six 2D fields, as in Eqn. (3.15), on a turbine inlet mesh at
the order of 10 000 points. A direct model would thus have some 100 000 degrees
of freedom which could not be reasonably interpreted physically. That is, why a
reduced-order model with only few parameters \( X \) is required. Contemporary ways
to reduce dimensionality in fluid flow problems are decompositioning techniques
such as proper orthogonal decomposition or dynamic mode decomposition (Schmid
[160]). These methods, however, are difficult to employ in the context of this work
due to the high degree of abstraction required in the interpretation of flow modes
and the large data base necessary. Instead, a modelling approach based on the
superposition of canonical flow components was chosen. The components are based
on mathematical functions that mimic a realistic flow field determined by the input
parameters \( X \).
The disadvantage of this approach is that the resulting interface will not necessarily
be a solution to the Navier-Stokes equations and therefore automatically impose
a modelling error. Also, the connection between the model parameters \( X \) and
the geometry of the combustor is unknown, in general. The uncertainties of the
model parameters to be imposed can therefore not be deduced from uncertainties
in combustor geometry, operating conditions, etc. but must be specified based on
characteristic data of contemporary combustor outlet traverses.
The main advantages of this approach are its efficiency, because no combustor CFD
is included in the process, and the possibility to directly manipulate the parameters
of interest (swirl position and strength, hot spot position, etc.) in the traverse in a
scientifically comparable manner.
4.1.2 Characteristic Features of Lean Burn Traverses

A typical lean burn traverse to be modelled is shown in Fig. 1.2 b). It is instructive to analyse characteristic 2D features of exit traverses first to motivate some of the details of the model. As explained in Section 2.1.3, a lean burn traverse typically consists of a strong swirling motion\(^1\) with large radial and circumferential flow angles, a deficit in pressure and temperature inhomogeneities.

With regard to the distribution of incidence at the CTI interface, it is important to note that annular sectors are largely different from linear cascades because of asymmetries induced by the combustor liner. An asymmetry in averaged swirl flow angle \(\alpha_{\text{tan}}\) at the LSTR can be recognised in Fig. 1.5 where the absolute value of \(\alpha_{\text{tan}}\) is larger in the upper half of the sector. Similar observations were made in the Trisector rig by Bacci et al. [5].

![Diagram](image)

**Figure 4.1:** Entrainment of flow from the hub into the swirling motion of different annular rigs (\(^1\) from Andreini et al. [3]) vs. a linear cascade (\(^2\) from Jacobi et al. [88])

The effect of the liner geometry on the radial flow angle \(\alpha_{\text{rad}}\) is even more pronounced. As can be seen in Fig. 4.1 for the three different cases on the left, a much greater portion of flow from the hub is *entrained* into the overall swirl crossing the interface with large radial flow angle \(\alpha_{\text{rad}}\) which does not occur at the casing. The entrainment also causes a local reversal of the swirl-induced circumferential flow direction at the hub (\(\alpha_{\text{tan}}\) locally changes its sign, not visible). This effect cannot be observed in symmetric, linear cascades, where the turbine inlet swirl field is almost symmetrical, as shown in Fig. 4.1 (right).

To the author’s knowledge such generalities cannot be stated with respect to the distribution of temperature, as it is determined by the geometry, aerodynamic patterns, injection and operating condition of the individual combustor. In the model, the temperature field is therefore represented by generalised radial gradients, due to wall cooling, and local hot spots, from the discrete locations of fuel injection.

---

\(^1\) The large scale swirl in the time mean of the flow may be accompanied by smaller vortices in between two adjacent swirl cores, as has been shown by Werschnik [180] for the LSTR. The effect of these smaller vortices is not considered in this work, although, in principle, this would be possible with the modelling approaches shown in the next sections.
4.2 Overall Parameterisation Scope and Strategy

4.2.1 Structure, Inputs and Outputs

The model described in the following is an extension of the version published by Schneider et al. [161]. The overall inputs and outputs of the model are displayed in Fig. 4.2. Information on the interface geometry (Geo), the operating conditions (OC) and a set of at least \( n = 36 \) input parameters \( X \) are provided to the model. From this information, the 2D distribution of all flow variables \( \Phi_{BC} \) required for an inlet BC, Eqn. (3.15), is computed on a numerical mesh. A grid size of \( [N_{rad} \times N_{tan}] = [200 \times 200] \) nodes has been found sufficient for a grid-independent generation of traverses with maximal local changes in total pressure due to grid refinement below 100 Pa. One iteration of the computation of \( \Phi_{BC} \) is depicted in the box, bounded by the red dashed line, in Fig. 4.2 and will be described in detail below.

The distributions of stagnation temperature \( T_t \) and swirl \( [u_{rad}, u_{tan}] \) are suited best to characterise a lean-burn CTI traverse. These quantities are therefore determined by the input parameter set \( X \). All other flow quantities are then derived from the distributions of \( T_t \) and \( [u_{rad}, u_{tan}] \) and information of the reference traverse. The iterative scheme is closed by the density field which is corrected until the maximal relative difference between the old and new density field is less than 0.1 %. The 2D flow field is then exported as a data file which can be imported by CFD solvers for use as inlet boundary condition. Postprocessing of the converged CFD results yields a set of output parameters

\[
Y = [\eta_{HPT}, \zeta_{NGV}, \Gamma, T_w, \text{Nu}, ...] .
\]  

(4.1)
If the CFD model $\eta$ can be regarded as a “deterministic black box” (epistemic uncertainties must be negligible, cf. Section 3.5.2), changes in outputs $Y$ can be expressed as a function of the input parameters $X$, according to Eqn. (3.23).

In the following, the notation $X_{\text{par}} \Rightarrow \Phi^{(f_i)}$ is used to indicate that a physical quantity $\Phi$ of field $f_i$ is determined by a subset of the input parameters $X$.

### 4.2.2 Constraints

As was shown in Chapter 3.3.3, the aero-thermal analysis of HPTs requires the scaling to common operating conditions in order to grant comparability. The generated flow fields are therefore scaled to obey an operating point defined by

$$T_t(r, \theta) = T_{t,DP}, \quad p_t(r, \theta) = p_{t,DP} \quad \text{and} \quad \int \rho(r, \theta)u_{ax}(r, \theta)r \, dr \, d\theta = \dot{m}_{DP}. \quad (4.2)$$

The generated flow fields are furthermore subject to the constraint of circumferential periodicity in order to allow for single-passage numerical computations as outlined in Section 3.2. Lastly, the generated flow profile must yield a valid numerical inflow condition, i.e., axial velocity must be positive everywhere on the CTI interface. This condition imposes a limit on the swirl strength, which can be imposed at the inlet of a decoupled simulation, at approximately $S = 0.3$. For swirl numbers above this value the low momentum in the swirl core may lead to problems with numerical stability at the interface. For the same reason, velocity very close to walls of the inlet plane is not set to zero (no-slip condition) but to a finite value.

### 4.3 Modelling of In-Plane Velocity Distribution

The 2D distribution of in-plane velocity components, i.e., radial velocity $u_{\text{rad}}(r, \theta)$ and circumferential velocity $u_{\text{tan}}(r, \theta)$, is superimposed from multiple components which are all forced to be circumferentially periodic. The modelling and computation of each component is described in the following.

#### 4.3.1 Swirl Field

Combustor exit swirl is modelled by a time-independent Lamb-Oseen type of vortex

$$\left[ d^{(\text{sw})}, D^{(\text{sw})}, \Gamma \right]_{\text{par}} \Rightarrow u^{(\text{sw})}(R) = d^{(\text{sw})} \frac{\Gamma}{2\pi R} \left[ 1 - \exp \left( -\frac{4R^2}{(D^{(\text{sw})})^2} \right) \right] \quad (4.3)$$

which describes an unbounded, viscous vortex by blending a solid body rotation in the core with diameter $D^{(\text{sw})}$ into a potential vortex in the outer region (Saffman [150]). A discussion of the model can be found in Appendix C.1. It offers three relevant parameters, the swirl direction $d^{(\text{sw})} = \pm 1$, the diameter of the viscous swirl

---

2 Mass flow averaging is applied for $T_t$ and $p_t$ as defined in Eqn. (3.16).
core $D^{(sw)}$ and the swirling strength $\Gamma$. In axisymmetric domains swirling strength is accounted for by the swirl number $S_0$. The integration in the definition of $S_0$, Eqn. (2.2), is done starting from the swirl centre in the direction of $R$ up to the domain boundaries. Since a HPT inlet traverse is not axisymmetric but a sector of a circle, there is no unique radial coordinate $R$ and domain boundaries do not define a fixed upper limit for the integration. In order to circumvent this problem, a pseudo-swirl number is defined, in the context of this work, $S^{(sw)}$:

$$S := \frac{\iint \rho u_{ax}(u^{(sw)} R) r \, dr \, d\theta}{(r_s - r_h) \iint \rho u_{ax}^2 r \, dr \, d\theta},$$

(4.4)

allowing for the calculation of $S$ in non-axisymmetric domains which is further discussed in Appendix C.1.2. The offset $[\theta^{(sw)}, r^{(sw)}]$ of the vortex centre from the centre of sector is defined by the two parameters $\Delta \theta^{(sw)}$ and $\Delta r^{(sw)}$. An overview of all parameters used for modelling and positioning the 2D swirl field is given in Fig. 4.3 (left). Note, that $R$ in Eqn. (4.3) and (4.4) is defined relative to the swirl centre and $r$ is defined relative to the machine axis.

**Figure 4.3:** Swirler vortex modelling: An axisymmetric vortex is generated from five parameters (left) and stretched by superposition of multiple vortices (right).

When examining flow fields behind realistic combustion chambers, it can be seen that the shape of the swirl core is not necessarily axisymmetric but may be stretched in circumferential direction (cf. Fig. 1.2 b). This is modelled by distributing $N_{sw}$ vortices at a distance of $D^{(sw)}/2$ along the circumference at the same radius $r^{(sw)}$ and superimposing their individually induced vector fields $u^{(sw)}(r, \theta)$ to that of a stretched vortex $u_{str}^{(sw)}(r, \theta)$. The distance of the outermost vortices is defined by the stretching parameter $\chi^{(sw)}$ such that

$$[r^{(sw)}, \theta^{(sw)}, \chi^{(sw)}] \xrightarrow{\text{par}} u_{str}^{(sw)}(r, \theta) = \sum_{i=0}^{N_{sw}-1} u^{(sw)}(r, \theta, \theta^{(sw)}, \chi^{(sw)})$$

(4.5)

where $\theta^{(sw)} = \theta^{(sw)} + \frac{2i - N_{sw} + 1}{4} D^{(sw)}$ and $N_{sw} = \left\lfloor \frac{D^{(sw)}/2}{\chi^{(sw)} r^{(sw)} \theta_{\text{pitch}}} \right\rfloor - 1$

$\chi^{(sw)}$ is a discrete parameter (all other parameters, except $d^{(sw)}$, are continuous).
as depicted in Fig. 4.3 (centre and right). The vortex stretching interferes with the computation of the swirl number $S^*$ which is detailed in Appendix C.1.2.

![Figure 4.4: The modelled vortex induces aperiodic velocities at the boundaries (blue arrows). Vortices from the neighbouring sectors cancel this aperiodicity (red arrows) but add additional, aperiodic contributions on the remote sides (black arrows). Therefore, all $N_{sec}$ vortices must be superimposed for periodicity.](image)

Periodicity of the boundary conditions is a necessary requirement for turbomachinery CFD in order to reduce the computational domain according to the blade count (cf. Section 3.2). As can be seen in Fig. 4.3, the sector modelled so far is not periodic. In order to enforce sector periodicity, the influence of *ghost vortices* from neighbouring passages must be accounted for and superimposed on the field $u_{str}^{\{sw\}}(r, \theta)$ in Fig. 4.3. However, as can be seen in Fig. 4.4, these additionally modelled vortices themselves induce non-periodic components at the respective remote boundary (black arrows). Only if the effect of all $N_{sec}$ ghost vortices around the circumference on the boundaries of the modelled sector is considered, i.e.,

$$u_{per}^{\{sw\}}(r, \theta) = \sum_{i=0}^{N_{sec}-1} u_{str}^{\{sw\}}(r, \theta, \left[r^{\{sw\}}_{gh,i}, \theta^{\{sw\}}_{gh,i}\right]),$$

where $\theta^{\{sw\}}_{gh,i} = \theta^{\{sw\}} + i\theta_{pitch}$ and $r^{\{sw\}}_{gh} = r^{\{sw\}}$, the problem becomes symmetric and the modelled sector $u_{per}^{\{sw\}}(r, \theta)$ is periodic.

### 4.3.2 Establishing Wall-Tangential Flow

The flow needs to smoothly follow the hub and shroud end walls in order to prevent a non-physical distribution of velocity or flow separation in these areas. This can be achieved by a two step procedure: First, the radial flow angle of the swirl field $u_{per}^{\{sw\}}(r, \theta)$ is set to zero at both end walls. Then, a radial velocity component is superimposed close to hub and shroud in order to set the flow angle to the geometric angle of the liner.

The first step can be achieved – similarly to enforcing periodicity – by placing

4 Note, that *stretched* ghost vortices are superimposed if $\chi^{\{sw\}} > 0$. 


mirrored ghost vortices outside the modelled flow sector. As shown in Fig. 4.5 (left), these vortices are placed at the same clocking position and wall distance as the main vortex but with reversed swirl direction, i.e.,

\[
u^{\text{sw}}_{\text{tw}}(r, \theta) = u^{\text{sw}}_{\text{per}}(r, \theta) + u^{\text{sw}}_{\text{per}}(r, \theta, \left[r^{\text{sw}}_g, d^{\text{sw}}_g\right]) + u^{\text{sw}}_{\text{per}}(r, \theta, \left[r^{\text{sw}}_{g,m,s}, d^{\text{sw}}_{g,m}\right]),
\]

where \( r^{\text{sw}}_{g,m,s} = \frac{1}{2}(3r_{hls} - r_{sh}) = \Delta r^{\text{sw}}, \) and \( d^{\text{sw}}_{g,m} = -d^{\text{sw}}. \) (4.7)

The radial components of the vortex in the sector and the ghost vortices cancel at the end walls and the vectors of the superposition \( u^{\text{sw}}_{\text{tw}}(r, \theta) \) point in axial direction.

Figure 4.5: A radial flow angle \( \alpha_{\text{rad}} = 0 \) at the end walls is established by superimposing the influence of mirrored vortices (red) outside the modelled sector (left). The flow angle \( \alpha_{\text{rad}} \) at the walls is then set to the geometric angle of the liner by superimposing a field of radial velocity which is uniform in circumferential direction and has a radial profile \( \tilde{f}(h_{\text{rel}}) \) (right).

The second step – turning the flow at the end walls to be wall-tangential – is done by adding two fields of radial velocity, \( u^{\text{ec}}_{\text{rad},hls}(r) \) to the swirl field \( u^{\text{sw}}_{\text{tw}}(r, \theta) \). In order to limit the effect of turning to the proximity of the walls, both fields must have a finite value at one wall and decrease to zero at the opposing one. Therefore, the added flow field must scale in radial direction with a function \( f(h_{\text{rel}}) \) that is one at \( h_{\text{rel}} = 0 \) and zero at \( h_{\text{rel}} = 1 \) where \( h_{\text{rel}} = \frac{r-r_h}{r_s-r_h} \). The shape of \( f(h_{\text{rel}}) \) in between must be flexible in order to adjust its shape to a real flow field. The requirements are

5 Functions \( f, g, h \) and quantities \( \phi \) that apply equally to the hub (index \( h \)) and shroud (index \( s \)) components are denoted by \( h_{\text{ls}} \) in the following.

6 The radial flow components, added to the swirl field, interfere with the swirl clocking position set by the parameter \( \theta^{\text{sw}} \). The parameters \( \left[u^{\text{ec}}_{r,hls}, \psi^{\text{ec}}_h, r^{\text{ec}}_h, \theta^{\text{sw}}\right] \) and \( \theta^{\text{sw}} \) are therefore interdependent. A corresponding radial misalignment, from the steps described in the next section, is prevented by superimposing a uniform, circumferential velocity field \( \Delta u_{\text{tan}} \).

4.3 Modelling of In-Plane Velocity Distribution
met by the function \( \tilde{f}(h_{\text{rel}}) \) that is shown in Fig. 4.5 (right) and defined in Appendix C.1.3. The function is defined by two parameters and scaled by a factor\(^7\) \( \hat{u}_{r,h|ls}^{(\text{ec})} \) which yields the flow field

\[
\left[ \alpha_{r,h|ls}^{(\text{ec})}, \psi_h^{(\text{ec})}, \gamma_h^{(\text{ec})} \right] \xrightarrow{\text{par}} u_{r,h|ls}^{(\text{ec})}(r) = \hat{u}_{r,h|ls}^{(\text{ec})} \tilde{f}_{h|ls}(h_{\text{rel}}).
\]

(4.8)

It is applied similarly to cross flows from the hub and the shroud.

### 4.3.3 Modelling Entrainment of Surrounding Flow

The steps described above yield periodic 2D distributions of flow angles that are tangential to the hub and shroud liner and may be used as HPT inlet conditions. However, the result does not yet fully match realistic flow conditions due to the effect of flow entrainment discussed in Section 4.1.2.

The missing flow components (Entrainment) are visualised in the top part of Fig. 4.6 by subtracting a Modelled swirl field (resulting from the described steps) from a simulated Realistic flow field.

In order to motivate the approach to modelling the entrained field, it is further divided into a circumferential and radial component which are both shown on the top right in Fig. 4.6. It can be seen that additional circumferential flow components occur close to the end walls in the proximity of the borders of the swirl core. They act in opposite direction as the modelled swirl field, thus “turning” the flow at the end walls such that it gets entrained symmetrically into the swirl. Additional radial components locally enhance the radial flow angle in the entrained region.

Both of these components can be sub-divided into a contribution from the hub and shroud, respectively, shown as contours of 2D modelled fields at the bottom right in Fig. 4.6. Each field is constructed from a distribution in \( \theta \)-direction \( g_{h|ls}(\theta^*) \) and a distribution in \( r \)-direction \( f_{h|ls}(h_{\text{rel}}) \) for the circumferential component, \( h_{h|ls}(h_{\text{rel}}) \) for the radial component. The shape of \( f_{h|ls}, g_{h|ls} \) and \( h_{h|ls} \) is illustrated at the bottom left in Fig. 4.6. The functions are derived from considerations discussed in Appendix C.1.4 and determined by parameters \( \Delta \theta^{(\text{fe})}, \zeta^{(\text{fe})} \) and \( \psi^{(\text{fe})} \) which adds 14 additional parameters to the overall model.

The 2D fields are computed from the product of both functions, each weighted with a scaling parameter \( \hat{u}_{\text{tan}|r|ls}^{(\text{fe})} \),

\[
\left[ \hat{u}_{\text{tan}|h|ls}^{(\text{fe})} \Delta \theta_{\text{tan}|h|ls}^{(\text{fe})}, \psi_{\text{tan}|h|ls}^{(\text{fe})} \right] \xrightarrow{\text{par}} u_{\text{tan}|h|ls}^{(\text{fe})}(r, \theta) = \hat{u}_{\text{tan}|h|ls}^{(\text{fe})} f_{h|ls}(h_{\text{rel}}) g_{h|ls}(\theta^*),
\]

\[
\left[ \hat{u}_{\text{rad}|h|ls}^{(\text{fe})} \Delta \theta_{\text{rad}|h|ls}^{(\text{fe})}, \psi_{\text{rad}|h|ls}^{(\text{fe})}, \zeta_{\text{h|ls}}^{(\text{fe})} \right] \xrightarrow{\text{par}} u_{\text{rad}|h|ls}^{(\text{fe})}(r, \theta) = \hat{u}_{\text{rad}|h|ls}^{(\text{fe})} h_{h|ls}(h_{\text{rel}}) g_{h|ls}(\theta^*).
\]

(4.9)

(4.10)

\(^7\) Instead of the radial velocity \( \hat{u}_{r,h|ls}^{(\text{ec})} \), the flow angle \( \alpha_{r,h|ls}^{(\text{ec})} = \tan \left( \hat{u}_{r,h|ls}^{(\text{ec})} / u_{ax,h|ls} \right) \) is defined as input parameter which is a geometric parameter of the liner and decoupled from the axial velocity.
4.3.4 Assembling the final In-Plane Vector Field

The three described 2D fields – \(u^{(sw)}\) for swirl, \(u^{(ec)}\) for end wall convergence and \(u^{(fe)}\) for flow entrainment – are assembled\(^8\) to the final in-plane velocity field

\[
\begin{align*}
    u_{\text{rad}}(r, \theta) &= u_{\text{rad}}^{(sw)}(r, \theta) + \max[u_{\text{rad}}^{(ec)}(r, \theta), u_{\text{rad}}^{(fe)}(r, \theta)] + \min[u_{\text{rad}}^{(ec)}(r, \theta), u_{\text{rad}}^{(fe)}(r, \theta)] \\
    u_{\text{tan}}(r, \theta) &= u_{\text{tan}}^{(sw)}(r, \theta) + u_{\text{tan}}^{(fe)}(r, \theta) + u_{\text{tan}}^{(ec)}(r, \theta),
\end{align*}
\]

(4.11)

where \(u_{\text{rad}}^{(ec|fe)}(r, \theta) \geq 0\) and \(u_{\text{rad}}^{(ec|fe)}(r, \theta) \leq 0\).

4.4 Modelling of Axial Velocity Distribution

As described in Section 3.3.2, there is a discontinuity in the distribution of velocity magnitude \(u\) inherent to the approach of decoupling the combustor and turbine

---

\(^8\) The radial components of \(u^{(ec)}\) and \(u^{(fe)}\) are not added but the maximal contribution of both fields is used in order to grant independence of their respective parameters.
simulations (velocity distributions at the CTI interface are different in coupled and decoupled simulations). Since velocity magnitude \( u \) is not prescribed as inlet BC \( \Phi_{BC} \), Eqn. (3.15), the axial velocity distribution \( u_{ax}^*(r, \theta) \) in the converged CFD solution is unknown prior to the simulation.

In the model, however, \( u_{ax}^*(r, \theta) \) is required to scale the mass flow-averaged inlet conditions of the turbine, Eqn. (4.2), and the swirl number \( S \), Eqn. (4.4). Thus, there is a “chicken-and-egg” situation because the distribution \( u_{ax}^*(r, \theta) \) is required to model CFD inlet conditions but there is no way to calculate \( u_{ax}^*(r, \theta) \), consistent with the rest of the traverse, without running CFD. Therefore, the model depends on the provision of an axial velocity field \( u_{ref, ax}(r, \theta) \) from an external reference as “starting point”. If this reference traverse is reproduced by the model, \( u_{ref, ax}(r, \theta) \) is equal to \( u_{ax}^*(r, \theta) \) and operating conditions can be scaled appropriately.

However, if the model is used to investigate deviations from the reference, CFD will yield a velocity distribution \( u_{ax}^*(r, \theta) \) different from \( u_{ref, ax}(r, \theta) \). In this case, \( u_{ref}(r, \theta) \) is used as an initial guess for the axial velocity distribution in the model but corrected by running a “dummy” CFD and using its \( u_{ax}^*(r, \theta) \) to generate a corrected traverse (the procedure is illustrated in Fig. 4.13).

### 4.5 Modelling of Total Pressure Distribution

The model must provide an inlet total pressure profile \( p_t = p + \frac{1}{2} \rho u^2 \) that is prescribed as boundary condition in \( \Phi_{BC} \). As was discussed in Section 2.2.3, the distribution of \( p_t \) is crucial for the formation of secondary flows in the NGV passage. From the steps described above, all velocity components are known but no information on static pressure \( p(r, \theta) \) is available (the density \( \rho(r, \theta) \) is computed in the last step from the, then known, pressure and temperature, Fig. 4.2). Although typical spatial variations\(^{10}\) of \( p \) across the interface are small, \( \Delta p / p = \mathcal{O}(0.01) \), compared to variations in Mach number, \( \Delta Ma = \mathcal{O}(0.1) \), they are not negligible (Appendix C.2.1) but a 2D pressure profile must be used in the model.

Similarly to \( u_{ax}(r, \theta) \), static pressure distribution from a reference \( p_{ref}(r, \theta) \) is used as initial guess in a correction loop and \( p^*(r, \theta) \) from a dummy CFD is used in the final traverse generation. If no reference field \( p_{ref}(r, \theta) \) is available, the pressure profile can be approximated from a momentum balance in radial direction as shown in Appendix C.2.2. The design operating pressure \( p_{t,DP} \) according to Eqn. (4.2) is set by adding a constant offset \( \Delta p_{DP} \) to the static pressure distribution \( p^*(r, \theta) \)

\[
p(r, \theta) = p^*(r, \theta) + \Delta p_{DP} \Rightarrow p_t(r, \theta) = p_{t,DP}.
\]  

\(^9\) If no reference \( u_{ax}^*(r, \theta) \) is available, a constant distribution of axial velocity \( u_{ax,0} \) is assumed which is scaled to the operating condition for \( \dot{m}_{DP} \), Eqn. (4.2).

\(^{10}\) Typical values are retrieved from Engine 3E inlet conditions (Fig. 5.18).
4.6Error Feedback in the Correction Loops of Axial Velocity and Pressure

The profiles of axial velocity $u^{\ast}_{ax}(r, \theta)$ and pressure $p^{\ast}(r, \theta)$, which are a result of CFD, and the imposed total pressure distribution $p_t(r, \theta)$, which is an input to CFD, are physically coupled. Deviations $\Delta u_{ax}$ and $\Delta p$ of the reference fields $u^{\{\text{ref}\}}_{ax}(r, \theta)$ and $p^{\{\text{ref}\}}(r, \theta)$ from the “real” traverse therefore cause a deviation in the modelled total pressure, because

$$p_t = (p^{\{\text{ref}\}} + \Delta p) + \frac{1}{2\rho} \left[ (u^{\{\text{ref}\}}_{ax} + \Delta u_{ax})^2 + u_{rad}^2 + u_{tan}^2 \right]. \quad (4.13)$$

This deviation in $p_t$ propagates through the CFD and feeds back to the model, because $u^{\ast}_{ax}$ and $p^{\ast}$ from CFD are used for correction. The correction loops for axial velocity and pressure are therefore not independent from the initial guesses $u^{\{\text{ref}\}}_{ax}$ and $p^{\{\text{ref}\}}$. In general, $\Delta u_{ax} \neq 0$ and $\Delta p \neq 0$, causing a persistent error in $p_t$, that cannot be corrected for a posteriori.

The model can therefore only produce reasonable results in the intended application if the effects of $\Delta p$ and $\Delta u_{ax}$ on the results $Y$ are negligible compared to those imposed by changing $u_{rad}(r, \theta)$, $u_{tan}(r, \theta)$ and $T_t(r, \theta)$ via the inputs $X$. A study on the influence of this effect can be found in Section 5.3.2. The implementation of the correction loop into the overall workflow is discussed in Appendix C.5.8.

4.7 Modelling of Total Temperature Distribution

The stagnation temperature field $T(r, \theta)$ is modelled by superimposing a background field $T^{\{\text{bg}\}}$ and an arbitrary number of hot spots $T^{\{\text{hs}\}}$ (referenced by index $i \in [1, 2, \ldots, n]$), as shown schematically in Fig. 4.7.

![Figure 4.7: Parameterised temperature profile as hull of a background profile and multiple hot spots. A reference field $T^{\{\text{ref}\}}$ is blended into the generated profile at the walls to account for circumferential non-homogeneity of wall film cooling.](image)

The background field is used to scale the mean temperature to operating conditions, Eqn. (4.2), and to account for radial inhomogeneity from combustor wall film cooling. Circumferential variations in the main flow are accounted for by the hot...
spots only. The superimposed overall temperature profile is the hull of the $n$ hot spots $T_i^{\text{hs}}$ and the background profile $T^{\text{bg}}$

$$T(r, \theta) = \max \left[ T_1^{\text{hs}}(r, \theta), T_2^{\text{hs}}(r, \theta), \ldots, T_n^{\text{hs}}(r, \theta), T^{\text{bg}}(r) \right]. \quad (4.14)$$

If the temperature distribution close to the wall is known from a reference solution $T^{\text{ref}}(r, \theta)$, it is blended into the temperature field $T(r, \theta)$ in a last step. This allows to better account for circumferential inhomogeneities from wall coolant flows which are not modelled explicitly.

### 4.7.1 Background Profile

The background profile is modelled as a circumferentially uniform field, dependent on span height $h_{\text{rel}}$ only. Its radial shape is set by a scaled power function $\Phi(h_{\text{rel}})$

$$[T^{\text{bg}}_{h_\text{s},w}, \psi^{\text{bg}}_{h\text{s}}] \xrightarrow{\text{par}} T^{\text{bg}}(r) = \left\{ \begin{array}{ll} \left( T_{\text{max}}^{\text{bg}} - T_{s,w}^{\text{bg}} \right) \Phi(h_{\text{rel}}) \psi_s^{\text{bg}} + T_{s,w}^{\text{bg}} & \text{if } h_{\text{rel}} \leq 0.5, \\ \left( T_{\text{max}}^{\text{bg}} - T_{h,w}^{\text{bg}} \right) \Phi(h_{\text{rel}}) \psi_h^{\text{bg}} + T_{h,w}^{\text{bg}} & \text{if } h_{\text{rel}} > 0.5, \end{array} \right. \quad (4.15)$$

where $\Phi(h_{\text{rel}}) = C_2 h_{\text{rel}}^2 + C_1 h_{\text{rel}} + C_0$. The coefficients $C_i$ are set, such that $\Phi(0) \approx 0$, $\Phi(0.5) \approx 1$ and $\Phi(1) \approx 0$. The wall temperatures $T_{h_\text{s},w}^{\text{bg}}$ and the exponents $\psi_{h\text{s}}^{\text{bg}}$ are four open parameters to set the end wall temperature and the thickness of a cold flow layer accounting for combustor wall cooling at hub and shroud, respectively (cf. Fig. 4.7, left). The peak temperature $T_{\text{max}}^{\text{bg}}$ of the background profile is not a free parameter but it is constrained by the mean temperature $\overline{T_i}$ according to Eqn. (4.2).

### 4.7.2 Hot Spots

Each hot spot $i$ is modelled as a normalised 2D Gaussian bell curve$^{12}$ $\tilde{f}(r, \theta) = f(r, \theta) / \max[f(r, \theta)]$ (Eqn. 3.19), weighted with a temperature $T_{i_{\text{max}}}^{\text{hs}}$

$$[r_i^{\text{hs}}, \theta_i^{\text{hs}}, T_i^{\text{hs}}, \sigma_{x,i}^{\text{hs}}, \sigma_{y,i}^{\text{hs}}, \phi_i^{\text{hs}}] \xrightarrow{\text{par}} T_i^{\text{hs}}(r, \theta) = T_{i_{\text{max}}}^{\text{hs}} \tilde{f}(r, \theta). \quad (4.16)$$

The bell curve $f(r, \theta)$ can be approximated by

$$f(r, \theta) = \exp \left[ -a_i \left( \theta_i^{\text{hs}} \right)^2 + 2b_i \theta_i^{\text{hs}} r_i^{\text{hs}} - c_i \left( r_i^{\text{hs}} \right)^2 \right] \quad (4.17)$$

where $a_i = \frac{\cos(\phi_i)^2}{2\sigma_{x,i}^2} + \frac{\sin(\phi_i)^2}{2\sigma_{y,i}^2}$, $b_i = -\frac{\sin(2\phi_i)}{4\sigma_{x,i}^2} + \frac{\sin(2\phi_i)}{4\sigma_{y,i}^2}$ and $c_i = \frac{\sin(\phi_i)^2}{2\sigma_{x,i}^2} + \frac{\cos(\phi_i)^2}{2\sigma_{y,i}^2}$. Similar to the swirl core, each spot $i$ is positioned at an offset from the centre of the sector by $\Delta r_i^{\text{hs}}$ and $\Delta \theta_i^{\text{hs}}$ and has an individual peak temperature $T_{i_{\text{max}}}^{\text{hs}}$, scaling parameters

$^{11}$ A Gaussian filter is additionally applied to prevent high spatial gradients in the resulting fields.

$^{12}$ All values of $\tilde{f}(r, \theta)$ below 0.2 are set to zero to prevent an interaction of the “tails” of the curve with the cold regions at the walls.
\(\sigma_{x,i}^{\text{hs}}, \sigma_{y,i}^{\text{hs}}, \) determining the size of the hot spot in two dimensions, and rotation
angle \(\phi_{i}^{\text{hs}}\). The implication of the individual parameters is displayed in Figure 4.7.

4.7.3 Accounting for Combustor Wall Film Cooling

The temperature close to the end walls in the CTI interface is strongly influenced
by residual coolant flow. The temperature profile is typically highly non-uniform in
this region because traces from individual combustor cooling holes can be identified.
Modelling these features individually would require a large number of parameters
and is beyond the scope of this work. Therefore, wall film cooling is accounted for
in the model using the following technique. A threshold temperature \(T^*\) is defined.
In regions with \(T_{t}^{\text{ref}} < T^* \) (close to the walls), the background profile \(T^{\text{bg}}\) from
Eqn. (4.15) is replaced with the reference field. The two fields are then linearly
blended into each other on a radial strip of \(\Delta h = 5\%\) span as described in Appendix
C.3. The result of this procedure is illustrated in Fig. 4.7 (right).

4.7.4 Temperature Scaling

The turbine’s inlet temperature needs to be scaled to fixed operating conditions
for comparability (cf. Section 3.3.3). In the model, this is achieved by adjusting
the maximal temperature of the background profile \(T_{\text{max}}^{\text{bg}}\), such that Eqn. (4.2)
is fulfilled, without changing the temperatures of the hot spots \(T_{i,\text{max}}^{\text{hs}}\) or the wall
temperatures \(T_{h|s,w}\).

This approach to scaling \(\overline{T}_t\) may fail if large hot spots with high temperature are
present because then the averaged temperature in the traverse cannot be sufficiently
reduced by a reduction of \(T_{\text{max}}^{\text{bg}}\). In this case, the temperature of at least one hot spot
\(T_{i,\text{max}}^{\text{hs}}\) needs to be lowered to match the specified operating point. The procedure is
illustrated for a realistic application in Fig. 5.20.

4.8 Modelling of Turbulence Distribution

The distribution of turbulence in the CTI traverse is highly dependent on the history
of the flow before the interface. It is therefore not possible to derive coherent
information on turbulent stresses just from information on the time-averaged flow
in the CTI traverse. Hence, the 2D distribution of turbulence is not modelled but a
circumferential average of the reference field is used as inlet condition if applicable.

In Section 5.3.2, the effect of an uncertainty in the mean intensity \(\overline{T}_u\) at turbine inlet
is investigated. Therefore, the circumferentially averaged reference profile \(T_u^{\text{ref}}(r)\)
is scaled by a constant \(C\) such that

\[
\overline{T}_u \overset{\text{par}}{\rightarrow} \frac{1}{m} \int \int \rho u_{ax} \left[ C T_u^{\text{ref}}(r) \right] r \, dr \, d\theta = \overline{T}_u. \tag{4.18}
\]
Tu thus serves as an additional input parameter to scale the turbulence intensity profile while preserving its radial distribution.

### 4.9 Summary of Model Parameters and Discussion of their Interdependence

Overall, the model contains at least $n = 36$ parameters for the velocity and stagnation temperature fields (depending on the number of modelled hot spots) as listed in Tab. 4.1. Parameters are divided into groups that scale the magnitude of certain effects or determine their position and shape.

**Table 4.1: List of Parameters for Combustor Exit Flow Model**

<table>
<thead>
<tr>
<th>Field and Component</th>
<th>Parameters</th>
<th>Scale</th>
<th>Position</th>
<th>Shape</th>
<th>$n$</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Velocity</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Inlet swirl</td>
<td>$S$</td>
<td>$\theta^{\text{sw}}$</td>
<td>$r^{\text{sw}}$</td>
<td>$\chi^{\text{sw}}$</td>
<td>$d^{\text{sw}}$</td>
</tr>
<tr>
<td>Endwall convergence</td>
<td>$\alpha_{\text{rad,h,s}}$</td>
<td>$\Delta \theta^{\text{fe}}_{\text{rad,h,s}}$</td>
<td>$\Delta \theta^{\text{fe}}_{\text{rad,h,s}}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Flow entrainment</td>
<td>$u^{\text{fe}}<em>{\text{rad,h,s}}$, $u^{\text{fe}}</em>{\text{tan,h,s}}$</td>
<td>$\Delta \theta^{\text{fe}}_{\text{rad,h,s}}$</td>
<td>$\Delta \theta^{\text{fe}}_{\text{tan,h,s}}$</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Temp.</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Background profile</td>
<td>$T^{\text{bg}}_{\text{h,s,w}}$, $r^{\text{hs}}$, $\theta^{\text{hs}}_i$</td>
<td>$\psi^{\text{bg}}_{\text{h,s}}$</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hot spot $i$</td>
<td>$T^{\text{hs}}_{i,\text{max}}$, $r^{\text{hs}}_i$, $\theta^{\text{hs}}_i$</td>
<td>$\sigma^{\text{hs}}<em>{x,i}$, $\sigma^{\text{hs}}</em>{y,i}$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The purpose of the flow model is to transfer these $n$ parameters into 2D HPT inlet conditions for UQ studies. As was discussed in Section 3.5.4, the number of required model evaluations $N$ in UQ studies grows quickly with dimension $n$ and 36 parameters cannot be realistically considered (a “truly” 36-dimensional MCS would require about 70 billion simulations).

For UQ, the parameters in Tab. 4.1 are therefore divided into 1st order parameters $\Phi_{1O}$ (dark grey cells in Tab. 4.1), which are varied, and 2nd order parameters $\Phi_{2O}$, which are not varied. The classification is done based on the physical relevance of the parameters in the design process.

However, not all 2nd order parameters can remain unchanged in the UQ process because the entrained flow is physically coupled with the swirl field13 as indicated in Fig. 4.6. A change in radial or circumferential position of the swirl field will evoke

---

13 A similar physical coupling of other parameters as well as a statistical dependence of some of the parameters is expected but not considered in this work due to a lack of the required data.
corresponding changes in the position of the entrained flow. Similarly, a coupling of swirl number \( S \) with the magnitude of entrained flow must be expected. Therefore, changes in the 1st order parameters of the swirl field \( \Delta \Phi_{1O} \) must cause changes in the dependent 2nd order parameters of entrained flow \( \Delta \Phi_{2O} \) (light grey cells in Tab. 4.1). The deviations in these parameters are linked by functions

\[
\Delta \Phi_{2O} = f(\Delta \Phi_{1O}), \quad (4.19)
\]

which are discussed in Appendix C.4. The functions are derived from the different measured and predicted swirl fields in the LSTR (Fig. 1.5) but used interchangeably for all cases shown in this work due to a lack of more general data.

4.10 Model Evaluation

4.10.1 Matching a Target Interface using Numerical Optimisation

The combination of model parameters \( X_{\text{opt}} \) that matches best a given reference field is determined by an optimisation algorithm or “optimiser”. This is an algorithm that, given a set of input parameters \( x \) each with an allowed range \([x_{\text{min}}, x_{\text{max}}]\) and a function \( f \) generating an output \( Y = f(X) \), provides the set of parameters \( x_{\text{opt}} \) that minimises \( Y \). Information on the genetic optimisation algorithm used in this work can be found in Appendix C.5.1. The matching is done separately for the flow angle/total pressure and temperature field of a given combustor turbine interface. The single objective fitness function \( f(X) \) to be minimised is the weighted sum

\[
Y = \sum_{k=1}^{K} C_k \Delta \Phi_k \text{ where } \Delta \Phi_k = \frac{\iint \widetilde{\Phi}_k \xi r \, dr \, d\theta}{\iint \xi r \, dr \, d\theta}, \quad \widetilde{\Phi}_k = \left| \Phi_k(r, \theta) - \Phi_k^{\text{ref}}(r, \theta) \right|_{\text{max}} \left[ \Phi_k^{\text{ref}}(r, \theta) \right], \quad (4.20)
\]

\( K \) is the number of flow fields \( \Phi_k(X) \) to be simultaneously optimised (1 for temperature, 3 for flow angles and total pressure), \( C_k \) is a weighting factor of the different fields and \( \xi \) determines the averaging procedure. In the results shown, area averaging is used, i.e., \( \xi = 1 \). In the matching of the flow angles and pressure fields with \( \Phi_k = [p_t, \alpha_{\text{rad}}, \alpha_{\text{tan}}] \) the respective weights are set to \( C_k = [0.5, 0.25, 0.25] \). Due to the linking of parameters, described in Section 4.9, only first order and independent second order parameters are open for the optimiser (as listed in Appendix C.5.2). The parameter vector \( X \) for the initial population is determined as best guess by the user before the automatic optimisation is started. About 30000 modelled traverses are typically evaluated per matching run.

4.10.2 Test Case LSTR

The model is evaluated by matching turbine inlet BCs obtained from combustor CFD (referred to as “datum” conditions), imposing the datum and matched conditions to the same CFD set-up and comparing the results from both traverses.

4.10 Model Evaluation
(a) Datum and matched quantities at ME01 (NGV inlet) as 2D contour and circ. averages

(b) Circumferentially averaged quantities at ME02 (NGV outlet)

(c) Comparison of film cooling effectiveness $\eta_{FC}$ at different axial cuts

Figure 4.8: Matching of a datum LSTR inlet traverse (derived from a coupled CFD at SwP clocking) in terms of aerodynamics at ME01 (a), ME02 (b) and cooling at the hub (c). The isothermal flowfield is determined by 14 independent parameters. Axial velocity $u_{ax}(r, \theta)$ and static pressure $p(r, \theta)$ from the datum field are used in the model. Standard deviations (a, right) indicate the variance from 30 repetitions of the matching.

In a first test case the approach is applied to the set-up of the LSTR as described in Section 3.4.1. The matched parameters $X$ are shown in Appendix C.5.5. The datum inlet conditions at SwL clocking are obtained from the coupled SAS simulation by Hilgert et al. [73] (the inlet conditions are shifted to the SwL clocking position). The swirl field is matched using the method described in the previous chapter where $u_{ax}(r, \theta)$ and $p(r, \theta)$ fields from the datum are used in the parameterised model, $T_i$ is set to a constant value (the rig is isothermal) and turbulent quantities $k$ and $\epsilon$ are prescribed as circumferential averages of the datum field.
The result of the matching process is shown in Fig. 4.8 a). The fields can be qualitatively matched well (left) but differences in circ. averaged flow angles of 5 to 10° occur from 0 to 20 % span for \( \alpha_{\text{rad}} \) and 60 to 100 % for \( \alpha_{\text{tan}} \) (right). The swirl field can be reproduced quantitatively only within the bounds of this model error. It can be seen that the variation of 30 different runs of the optimiser is well below the modelling error (grey band in Fig. 4.8 a, right), i.e., the optimisation process shows an adequate repeatability. In order to estimate how the differences between match and datum affect the resulting flow solution, simulations are conducted using both inlet conditions. The results are shown in Fig. 4.8 b) and c). It can be seen that the differences in circumferentially averaged quantities are smaller than the difference between CFD and experiment. Moreover, the distribution of \( \eta_{\text{FC}} \) from the datum inlet condition in Fig. 4.8 c) is matched with good agreement which is well below the difference between the datum CFD and the measurements at all evaluated axial positions.

4.10.3 Test Case RRD-HPT

The second test case used for validation is a high pressure turbine stage, representative for a current engine design of the industrial partner RRD, at cruise condition. For reasons of confidentiality it is referred to as “RRD-HPT”. This case has a simpler geometry than that of the LSTR NGV1 because RIDN and film cooling as well as complex geometrical features are not included in the numerical model (seal and leakage flows from cavities between the NGV1 and rotor row are included) allowing for a structured mesh generation and a computationally efficient numerical model.

![Figure 4.9: Definition of clocking position of the 2D inlet condition with respect to the NGV vanes (left) and comparison of the datum and matched boundary condition (right). The 2D flow field is indicated by vectors, red zones indicate the position of hot spots in the original and modelled traverse.](image)

4.10 Model Evaluation
The stage is not choked at the investigated operating point. A mass flow exit BC is applied in this case (which is uncommon for HPT simulations) in order to suppress changes in capacity and to enable a better comparability of efficiency changes resulting from the inlet conditions. The radial shape (not magnitude) of static outlet pressure is constrained using a profile from a multi-stage simulation. The inlet traverse, displayed in Fig. 4.9, is matched by the model assuming a constant distribution of axial velocity instead of a reference profile. Also, the error in axial velocity in the scaling of operating conditions is not corrected for.

In a first step differences in efficiency $\eta$, NGV pressure loss coefficient $\zeta_{NGV}$ and stage capacity $\Gamma$ are compared between the datum and matched inlet BC. In order to allow for a comprehensive comparison, inlet conditions are not only compared at a single clocking position, but they are clocked across an entire pitch at eight positions. Results are shown in Fig. 4.10. In this section, the agreement between datum and matched traverse is discussed only, a more detailed discussion on the relation of stage efficiency with swirl clocking position can be found in Section 5.2.2.

A qualitatively good match can be seen for the NGV pressure loss curve, $\zeta_{NGV}$, which has an almost constant offset between datum and match at all clocking position. The offset is assumed to occur mainly due to a difference in $T_{t,\text{in}}$ between datum and match, as shown on the right in Fig. 4.10. The difference in inlet total pressure is an order of magnitude smaller and mass flow differences are negligible as they are fixed by the outlet BC.

The curves of isentropic efficiency $\eta$ show a similar trend, however, the offset is not as constant as for $\zeta_{NGV}$. Differences between the curves are assumed to result from varying ability to match distribution of temperature at NGV1 outlet at different clocking positions which is shown in Appendix C.5.4. As was discussed in Section 2.4.1, differences in NGV outlet temperature distribution cause a variation in rotor incidence and thus affect rotor losses.

In summary, it can be stated that distributions of inlet flow angles and stagnation
pressure, affecting NGV1 pressure loss\(^{14}\), are matched in good agreement. The matching of inlet temperature, that has an additional effect on efficiency \(\eta\), is worse which can be seen in the varying offset of the \(\eta\) curves.

![Figure 4.11: Comparison of Nusselt number \(\nu\) at the turbine walls using datum inlet conditions from combustor CFD and the parameterised, matched inlet conditions at four different clocking positions. The reference temperature \(T_{\text{ref}}\) was set constant at approx. 60% of \(T_{\text{t,in}}\).](image)

The distribution of Nusselt number \(\nu\), Eqn. (2.16), on the vane surfaces and hub end wall is shown in Fig. 4.11. A quantitative comparison of the distribution of \(\nu\) over the surface area can be found in Appendix C.5.7. Increased heat transfer can be recognised at the vane LE and towards the TE where boundary layers are thin due to the impingement in the stagnation region and the flow acceleration, respectively. Low heat transfer occurs in the front part of the hub below the region of high radial flow angles (cf. Fig. 4.9) and along a streak at the PS of the left vane. This region of low \(\nu\) is due to the induced vortex pair at the vane LE, shown in Fig. 2.8, which drives fluid away from the wall and thus locally thickens the boundary layer. It can be seen that the structure moves radially downwards with positive clocking of the 2D inlet conditions. The phenomenon will be discussed in detail in the first part of the next chapter.

The qualitative agreement of predictions of \(\nu\) between the datum and matched traverse is good. Regions of relatively high and low \(\nu\), especially those from the induced LE vortex, agree well for most clocking positions. The largest differences can be identified at the LE and PS of the right vane at the 3/4 pitch clocking position. At this clocking position additional vorticity is generated by the modelled entrained flow interacting with the vane which is attributed to the modelling assumptions discussed in Section 4.3.3. The quantitative comparison in Fig. 4.11 (bottom) shows that the largest differences occur for low \(\nu\) at the hub at all clocking positions.

\(^{14}\) Note that, according to the substitution principle (Section 2.4.1), the inlet stagnation temperature does not have and effect on NGV1 secondary flows.

4.10 Model Evaluation

95
These differences correspond to the low Nu in the region of flow entrainment, visible in Fig. 4.11 (top), which is more pronounced in the model.

![Figure 4.12](image)

**Figure 4.12:** Comparison of normalised adiabatic wall temperature $T_w/T_{\text{in}}$ using datum inlet conditions from combustor CFD and the matched, parameterised inlet conditions at four different clocking positions.

The temperature distribution on the vane surfaces and hub end wall is shown in Fig. 4.12. It can be seen that the qualitative features, characterising the heat load on the vanes at the different clocking position, are represented by the model. Nonetheless, large differences are visible on the right vane at the $1/4$, $1/2$ and $3/4$ pitch clocking position. These are assumed to result mainly from mismatches in the temperature traverse, as the agreement of Nu is better at these positions. Additional test cases with other inlet traverses are shown in Appendix C.5.6.

### 4.10.4 Summary of Combustor Exit Flow Model Evaluation

In summary, it can be stated that the iso-thermal flow field within the LSTR NGV1 row with inlet swirl can be represented by the combustor exit flow model with errors that are below the modelling error of CFD if $u_{\text{ax}}(r, \theta)$ and $p(r, \theta)$ from the datum field are used in the model.

The overall performance of the RRD-HPT stage is qualitatively reproduced by the modelled inlet BC at different clocking positions. The best and worst clocking positions of the BCs in terms of stage efficiency and NGV pressure loss can be identified with a maximal error of $1/8$ of a pitch. Heat transfer data show that the local influence of secondary flows on boundary layers can be reproduced by the parameterised BCs. Wall temperature distributions show a worse agreement with significant differences at some of the investigated clocking positions. Comparison of temperature contours of this uncooled set-up, however, can be regarded as a “worst case” estimation because, as will be shown in Section 5.4.3, differences in wall temperature distribution are evened out in the presence of RIDN and vane film cooling. Anyway, the observed differences indicate that it is necessary to evaluate the representation of wall temperatures by the matched BCs prior to conducting UQ.
analyses in the next chapter.
As a third test case, the engine 3E HPT inlet traverse is discussed in Section 5.4.3, showing an agreement qualitatively equal to the results in this section.

### 4.11 Integration into an Uncertainty Quantification Workflow

The integration of the presented model into a UQ workflow is shown in Fig. 4.13.

**Parameter matching**: A set of model parameters $X$ is fed to the model in order to create a 2D HPT inlet traverse. The matched parameters, that represent best the reference traverse, are determined by coupling the model with an optimiser in an iterative loop.

**Sampling**: A subset of the matched parameters is uncertain. The uncertainties must be prescribed by PDFs (blue). A sampling procedure yields $N$ sets of discrete parameter combinations $X$. A traverse is created for each sample.

**CFD and Postprocessing**: The CFD is run in two steps in order to correct the assumed velocity and pressure field. A set of output parameters $Y_j$ is derived from each simulation. All $N$ sets are post-processed in order to derive their statistical moments (red).

**Figure 4.13**: Uncertainty quantification workflow: ① A reference traverse is parameterised by matching a set of input parameters $X$ using a numerical optimiser. ② From these matched parameters $N$ sets of samples are created using given PDFs of all $X$. ③ Each sample set is then used to generate a traverse and run a CFD calculation which is split up into an initial and a final run to correct for influences of the distribution of $u_{ax}$ and $p$. ④ The results $Y$ are extracted from all $N$ simulations and used to compute the PDFs of $Y$. 

4.11 Integration into an Uncertainty Quantification Workflow 97
The inputs to this procedure are a 2D reference traverse (from combustor CFD, for instance) and the PDFs of the uncertain parameter set \( X \) defining the traverse in the model. The procedure yields the PDFs of the uncertain output parameters \( Y \) using the UQ methods introduced in Section 3.5. One iteration of the axial velocity and pressure correction process, discussed in Appendix C.5.8, is implemented.

The following list summarises explicit and implicit assumptions involved in the uncertainty quantification process proposed in Fig. 4.13:

- **Distribution of velocity** (→ Section 4.4, Appendix C.5.8): Changes in the inputs \( X \) are assumed to exert negligible changes in the distribution of velocity magnitude.

- **Traverse model** (→ Section 4.2 to 4.8): The model components presented in the first part of this chapter are assumed to sufficiently represent turbine inlet traverses with negligible errors.

- **Parameters varied** (→ Section 4.9): Only first order parameters determining the HPT inlet traverse are changed in order to investigate their effect on turbine performance. The effect of second order parameters is assumed negligible.

- **Flow entrainment at off-design conditions** (→ Section 4.9, Appendix C.4): The shape of the entrained flow profiles is assumed independent of the position and shape of the inlet swirl core. The relation of magnitude of entrained flow with swirl number \( S \) at the LSTR, used to link first order and second order parameters, is assumed representative for other cases.

- **Independence of combustor wall cooling and hot spot position** (→ Section 4.7): Footprints of combustor wall cooling in the CTI traverse are assumed independent of changes in hot spot position and velocity field.

- **Distribution of turbulence** (→ Section 4.8): The distribution of turbulence is taken as a circumferential average of the reference traverse and assumed independent of changes in temperature and velocity.

- **Velocity boundary layer at inlet** (→ Section 4.2.2): The influence of a velocity boundary layer at turbine inlet is neglected in the modelling in order to avoid numerical instabilities at the inflow boundary.

- **Deterministic CFD model** (→ Section 3.5.2): The CFD model is assumed deterministic, i.e., numerical errors and epistemic uncertainties due to discretisation, physical modelling, turbulence modelling, etc. are assumed to be decoupled from the aleatory uncertainties imposed by the inlet conditions.

- **Decoupling of turbine from combustor and location of inlet boundary** (→ Section 3.3.1): The location of the inlet boundary is assumed sufficiently far upstream to exert a realistic influence on the turbine. The upstream effect of turbine...
vanes on the combustion process and turbine inlet conditions is assumed negligible.

- **Distribution of model parameters** (→ Section 3.5): The PDFs of model parameters $X$ estimated or obtained from available test data are assumed representative for the real engine case.

- **Dependence of model parameters** (→ Section 3.5): All model parameters $X$ are assumed statistically independent in the analysis because their dependency is unknown. In particular, the variance in position and magnitude of hot spot and swirl are assumed to be independent.
5 Results and Discussion

5.1 Structure of this Chapter

This chapter consists of two parts. In the first part, effects of parameters \( \mathbf{X} \) determining the inlet swirl on vane aerodynamics under iso-thermal conditions are investigated using a generic inlet traverse with the aim to derive a comprehensive understanding of the effect of variations of inlet swirl on secondary flows in the NGV passage. In the second part, uncertainty propagation of the inlet conditions of an iso-thermal rig with inlet swirl (LSTR) and a realistic HPT set-up with swirl and temperature non-homogeneity (E3E) are analysed. According to the substitution principle (cf. Section 2.4.1) the streamline pattern of isentropic flow through a stator passage is not affected by changes of the inlet stagnation temperature profile. Therefore, the findings regarding aerodynamics of part one, with a homogeneous inlet temperature, are transferable to the realistic case with a non-uniform inlet temperature in part two.

5.2 Losses and Vane Secondary Flows Induced by Inlet Swirl

5.2.1 Numerical Set-up and Inlet Conditions

The geometry and numerical set-up of the RRD-HPT, as described in Appendix C.5.3, is used for the analyses in part one of this chapter. A generic inlet traverse with swirl and a homogeneous temperature distribution is applied (Fig. 5.1). Note, that in most simulations a mixing plane (cf. Section B.2) is applied. That is, only circumferential averages of NGV1 exit flow are passed to the rotor and unsteady blade row interaction is not resolved. However, the comparison with unsteady calculations, reported in Appendix D.1.3, indicates that transient blade row (TBR) effects do not substantially alter the qualitative findings from steady CFD. A homogeneous temperature distribution is applied in order to prevent problems with mass flow averaging in efficiency calculations (cf. Section 3.3.2). A generic swirl topology at a relatively high swirl number, \( S = 0.2 \), is chosen for this study in order to provoke visible, swirl-induced effects in the stator. As the field is generic, no reference fields \( u^{\text{ref}}_{\text{ax}}(r, \theta) \) and \( p^{\text{ref}}(r, \theta) \) can be used in the model. Hence, axial velocity is assumed constant in the initial generation of the traverses. The initial distribution of static pressure is approximated by a radial momentum balance as described in Appendix C.2.2. Dependent secondary parameters \( \Phi_{20} \) are determined from the procedure described in Section 4.9 and all simulations are iterated once,
as shown in Fig. 4.13, in order to correct $u_{ax}(r, \theta)$ and $p(r, \theta)$. Thus, inlet swirl numbers $S$ at different clocking positions are made comparable best possible.

![Swirl Clocking Position at Inlet](image_url)

**Figure 5.1:** Definition of the clocking position of inlet swirl with respect to the NGV vanes $\theta_{sw}^\mathrm{[sw]}$, swirl direction and visualisation of the topology of the inlet swirl field. The traverse shown corresponds to a swirl number of $S = 0.2$.

As shown in Fig. 5.1, the direction of inlet swirl is referred to as positive if it is oriented with the rotation of the rotor, otherwise negative. The vanes labelled $\mathbb{L}$ and $\mathbb{R}$ in Fig. 5.1 are referred to as left and right vane, respectively, in the following.

### 5.2.2 Stage Efficiency and NGV Pressure Loss

#### Effect of Swirl Number

First, the influence of the first order parameters in Tab. 4.1 determining the swirl field is investigated by means of MCS of which two runs are conducted. Tab. 5.1 shows the varied parameters and their distributions producing variations of the traverse in Fig. 5.1 at positive swirl direction. Parameters are sampled uniformly within their maximal limits using LHS. Traverses with swirl below $S = 0.1$ show small deviation from datum conditions, the upper limit $S = 0.3$ is set to prevent excessive backflow at the domain inlet. Limits of $\Delta r_{sw}^\mathrm{[sw]}$ and $D_{sw}^\mathrm{[sw]}$ are estimated to represent maximal physical variations.

Results of all conducted simulations are filtered for numerical quality before evaluation and interpretation. Simulations are not considered if imbalances of the solved equations are above $0.01\%$ of their maximal value, residuals are above $1 \times 10^{-4}$, efficiency $\eta$ varies by more than $0.1\%$ in the last 50 iterations, inlet total pressure is more than $200\,\text{Pa}$ off design conditions (variations in inlet temperature are de facto zero) or backflow takes place at more than $5\%$ of the inlet plane. The latter posed the most critical criterion and simulations with $S > 0.25$ could not be used for evaluation. Backflow is handled by the solver by locally setting the inflow velocity to zero which could not be avoided in some of the simulations with $0.2 < S < 0.25$. 

102
Table 5.1: Parameters and Distributions of the Conducted Monte Carlo Runs

<table>
<thead>
<tr>
<th>Run</th>
<th>Parameters</th>
<th>Limits</th>
<th>Distribution</th>
<th>Samples</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Swirl clocking $\Delta \theta^{\text{sw}}$</td>
<td>$\in [-1/2, +1/2]$ pitch</td>
<td>uniform</td>
<td>100 (LHS)</td>
</tr>
<tr>
<td></td>
<td>Swirl number $S$</td>
<td>$\in [0.1, 0.3]$</td>
<td>uniform</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Swirl clocking $\Delta \theta^{\text{sw}}$</td>
<td>$\in [-1/2, +1/2]$ pitch</td>
<td>uniform</td>
<td>160 (LHS)</td>
</tr>
<tr>
<td></td>
<td>Swirl number $S$</td>
<td>$\in [0.1, 0.3]$</td>
<td>uniform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Radial position $\Delta r^{\text{sw}}$</td>
<td>$\in [0, 0.1H]$</td>
<td>uniform</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Core diameter $D^{\text{sw}}$</td>
<td>$\in [0.1H, 0.3H]$</td>
<td>uniform</td>
<td></td>
</tr>
</tbody>
</table>

Results of the study are shown in Fig. 5.2 as a function of inlet swirl number $S$ on the abscissa since it has a dominant influence on stage efficiency within the limits investigated. It is emphasised that a mass flow outlet condition is set in order to keep all cases at constant stage capacity $\Gamma$ and averaged inlet total temperature and pressure are scaled to the same operating conditions. To examine the effect of the outlet condition, run 1 is repeated using a fixed pressure outlet condition which leads to changes in $\Gamma$ as shown in Appendix D.1.1. The efficiency deficit $\Delta \eta$ is smaller in this case but shows the same qualitative trends.

The following conclusions can be drawn from the results in Fig. 5.2 a): Increasing inlet swirl number $S$ increases NGV pressure loss $\zeta_{\text{NGV}}$, Eqn. (2.9), and decreases stage efficiency $\eta$, Eqn. (2.7), non-linearly. In the considered (large) range of swirl numbers $S$, there is a strongly negative correlation between $\zeta_{\text{NGV}}$ and $\eta$ of $\rho(\zeta_{\text{NGV}}, \eta) = -0.96^\circ$. That is, decreases of efficiency observed in this study can be assumed to be caused primarily by a pressure loss in the NGV. As can be seen by the colouring from blue to red, the clocking position has a clear effect on both pressure loss and efficiency where positive clocking towards the LE of the right vane (red points) is beneficial.

The additional consideration of variations of the swirl core radial position $\Delta r^{\text{sw}}$ (indicated by marker shape in Fig. 5.2 b) and diameter $D^{\text{sw}}$ (indicated by marker size) in run 2 increases scatter of the data. An effect of the clocking position can still be observed, however, less clearly. Correlation coefficients of $\Delta r^{\text{sw}}$ and $D^{\text{sw}}$ with $\eta$ are not significant because they are overshadowed by the strong correlation of $\eta$ with $S$. Nevertheless, Fig. 5.2 b) implies a positive effect of $D^{\text{sw}}$ on $\eta$, i.e., efficiency increases with swirl core diameter. This relation can be explained by changes in the inlet pressure profile: Smaller cores cause larger gradients of the inlet pressure profile which result in a more stable swirl structure throughout the stator passage. Note, that the initial inlet total pressure profile is generated based
on the assumptions of a radial momentum balance (Appendix C.2.2) and a uniform inlet velocity.

Figure 5.2: Efficiency deficit at increasing swirl number (positive rotation). The colour of marker faces indicates $\Gamma$, the colour of marker edges indicates $\Delta \theta^{\text{sw}}$. In the right figure the size of markers indicates $D^{\text{sw}}$ and swirl cores with positive/negative radial shift $\Delta r^{\text{sw}}$ are displayed by triangles pointing upwards ($\bigtriangleup$)/downwards ($\bigtriangledown$). A reference is provided by the dashed line which is a fit through real engine data from Schmid [157].

Effect of Swirl Clocking and Direction

Apart from $S$, the results of the previous section indicate a correlation of stage efficiency with inlet swirl clocking position $\theta^{\text{sw}}$. In order to further investigate into this relation, parameter studies are conducted with inlet swirl at 16 clocking positions $-\frac{1}{2}$ pitch $\leq \Delta \theta^{\text{sw}} < +\frac{1}{2}$ pitch at fixed swirl numbers $S$ of 0.1 and 0.2. Figure 5.3 shows the efficiency deficit $\Delta \eta$ as well as deficits in NGV pressure loss $\Delta \zeta^{\text{NGV}}$ for each clocking position. At $S = 0.2$ the swirl core passes the right vane’s LE between the positions $\Delta \theta^{\text{sw}} = 3/16$ and $4/16$ which is indicated by a vertical bar in the plots. Plots at the bottom of Fig. 5.3 show the respective curves at a swirl number of $S = 0.2$ for opposing orientations of inlet swirl (positive and negative

---

1 NGV pressure loss is reported both as $\zeta^{\text{NGV}}$, based on NGV exit pressure upstream of the mixing plane, and as mixed-out version $\zeta^{\text{NGV,mix}}$, based on the exit pressure downstream of the mixing plane. Therefore, in comparison to $\zeta^{\text{NGV}}$, the coefficient $\zeta^{\text{NGV,mix}}$ additionally includes mixing losses at the row exit.

2 Due to the vanes’ potential field, the swirl core trajectory is deflected before impinging the vanes. The clocking position where the swirl core impinges the LE (being split into a PS and SS leg) is therefore not aligned with the position where the inlet swirl passes the axial projection of the LE as shown in Fig. 5.1.
as defined in Fig. 5.1). A curve fit and interpolation of the data in Fig. 5.3 can be found in Appendix D.1.4. The SST-$k$-$\omega$ model is used in the simulations. The influence of turbulence modelling on the results is discussed in Appendix D.1.2.

The conclusions of Fig. 5.2 regarding swirl clocking can be found again in Fig. 5.3: A non-linear dependency of $\eta$ and $\zeta_{NGV}$ on swirl number $S$ as well as a beneficial behaviour for $\Delta \theta^{(sw)} > 0$ can be identified. Curves of $\Delta \eta$ and $\Delta \zeta_{NGV}$ resemble a sine wave with a single, distinct maximum and minimum which are approximately $\frac{1}{2}$ pitch apart. Comparing the curves of $\Delta \eta$ and $\zeta_{NGV,mix}$, it can be seen that the locations of maximal $\Delta \eta$ correspond to the minima in $\zeta_{NGV,mix}$ and vice versa (with maximal deviations of $\frac{1}{16}$ pitch). The deviations in efficiency resulting from inlet swirl are thus mostly due to additional entropy generation in the NGV row which is about twice as high as in the rotor (not shown). The worse agreement in the extrema of $\Delta \eta$ and the un-mixed pressure loss $\zeta_{NGV}$ shows, that NGV exit mixing plays a relevant role in entropy generation due to inlet swirl.

At the clocking position of maximal $\eta$ the swirl core is split into SS and PS legs by the vane LE (illustrated in Fig 5.8 c). For $S = 0.2$, this happens when the axial projection of inlet swirl is slightly clocked towards the PS of the right vane. For a smaller $S$ or reversed swirl direction the optimal clocking position moves by $+\frac{3}{16}$
The plots at the bottom of Fig. 5.3 indicate that $\Delta \eta$ and $\Delta \zeta_{\text{NGV}}$ show a similar behaviour if the inlet swirl rotation direction is reversed. A phase shift (between $1/16$ and $3/16$ pitch) of the curves is noticeable and efficiency, averaged over all clocking positions, is approximately $-0.1\%$ lower for negative swirl. This deficit, however, is not due to viscous entropy generation, but results mainly from a difference in transferred power $P$ caused by an overall smaller circumferential flow component $C_u$ at NGV exit (cf. Fig. 2.4) in agreement with the argumentation by Schmid [157].

**Figure 5.4:** Rotor-relative incidence $\Delta \beta_{\tan}$ (with respect to the axial inflow case, positive values denote positive incidence) and rotor entropy loss coefficient $\zeta_s$ for different clocking positions $\Delta \theta^{\text{sw}}$ and direction $d^{\text{sw}}$ of inlet swirl.

The distributions of circumferentially averaged, rotor-relative incidence $\Delta \beta_{\tan}$ (with respect to the reference case with axial inflow) at the stator-rotor interface and entropy loss coefficient $\zeta_s$, Eqn. (2.9), at three radial positions are displayed in Fig. 5.4 as a function of inlet swirl clocking position $\theta^{\text{sw}}$ for both positive and negative swirl direction. Rotor incidence at midspan is largely altered if inlet swirl is clocked from the passage ($\theta^{\text{sw}} < 0$) to the LE. Changes in $\Delta \beta_{\tan}$ at 50 % span occur similarly for both swirl orientations whereas the incidence situation at 10 % and 90 % span is qualitatively inverted which is explained in Section 5.2.3. At 50 % and 90 % span increased incidence qualitatively corresponds with increased losses in the rotor at most of the inlet swirl clocking positions. At 10 % span this correspondence cannot be observed due to the interaction with rotor wheel purge flow injected downstream of the stator-rotor interface.

### 5.2.3 Swirl Induced Radial Pressure Gradients and Secondary Flow Migration

It was shown in the previous section that the direction of inlet swirl affects the radial position of rotor incidence and thus rotor losses. In this chapter, the mechanisms responsible for radial swirl migration through the stator passage are investigated. As was shown in previous investigations (Schmid [157], Werschnik [180], Insinna et al. [79]), inlet swirl causes a local incidence that loads or unloads the vanes at different span heights depending on its sense of rotation. The deficit in vane lift, i.e.,
the integrated pressure difference between PS and SS with respect to axial inflow at constant span height at the left and right vane (L and R in Fig. 5.1) is shown in Fig. 5.5 for the two MCS studies introduced in Tab. 5.2. It can be seen that if inlet swirl is clocked towards the LE of the left vane (blue points) the left vane has a lower lift at 80% span and a higher lift at 20% span. The situation is vice versa for the right vane. This observation is in accordance with the fact that positive inlet swirl induces negative incidence (aerodynamically unloading the vane) towards the shroud and positive incidence (loading the vane) towards the hub.

![Graphs showing lift deficit at increasing swirl number (positive rotation). The colour of markers indicates Δθ^{sw}. In the right figure, the size of markers indicates D^{sw} and swirl cores with positive/negative radial shift Δr^{sw} are displayed by triangles pointing upwards (Δ)/downwards (∇).](image)

(a) Run 1: Δθ^{sw} and S varied  
(b) Run 2: Δθ^{sw}, S, Δr^{sw} and D^{sw} varied

Figure 5.5: Lift deficit at increasing swirl number (positive rotation). The colour of markers indicates Δθ^{sw}. In the right figure, the size of markers indicates D^{sw} and swirl cores with positive/negative radial shift Δr^{sw} are displayed by triangles pointing upwards (Δ)/downwards (∇).

As described by Jacobi et al. [88], the local blade loading shifts the stagnation line at the vane’s LE towards the SS or PS forcing the flow to accelerate towards
the respective opposite side (Section 2.4.1). Negative incidence therefore causes an acceleration of the flow around the LE towards the PS side, positive incidence towards the suction side. This, in turn, causes opposing radial pressure gradients at SS and PS because incidence due to inlet swirl always acts in opposite directions at hub and shroud. The direction of the gradient depends on the inlet swirl direction as shown in the centre of Fig. 5.6.

![Figure 5.6](image)

**Figure 5.6:** Static pressure at four locations indicated in the central column at both positive (top row) and negative (bottom row) direction of inlet swirl, clocked between ±1/2 pitch from the vane LE. Charts on the left show absolute pressure, normalised with mean inlet total pressure, charts on the right show pressure relative to the axial reference case in order to isolate the effect of inlet swirl.

In order to confirm and quantify the effect for the investigated geometry, the parameter studies shown in Fig. 5.3 are evaluated with respect to the local static pressure at the four locations indicated by triangles in Fig. 5.6 for different clocking positions of inlet swirl. The right vane (as defined in Fig. 5.1) is evaluated. In the left part of the figure, a variation of pressure with $\theta^{(sw)}$ at the four locations is visible which is approximately inverted for reversed swirl orientation.

The vanes of the investigated geometry are not prismatic and hub and shroud end walls are not contoured symmetrically. Therefore, a radial pressure gradient already exists in the axial reference case. In order to isolate the effect of inlet swirl, pressures are corrected for this gradient by showing the difference to the reference case in plots on the right of Fig. 5.6.

The following observations can be made: The direction of the radial pressure gradient at the different clocking positions is in accordance with expectations due to the local blade loading from inlet swirl. Clocking swirl with positive rotation towards the SS of the right vane $\theta^{(sw)} \approx 1/2$, for instance, imposes a radial pressure gradient at the PS driving fluid to the shroud and a pressure gradient at the SS
driving it to the hub (indicated by grey and black arrows in the central column). The pressure gradients are reversed for negative swirl direction. The effect manifests more symmetrically and with much larger magnitude (one order) at the SS. This difference between PS and SS is assumed to be dependent on the geometry of the vane’s LE.

The pressure gradient vanishes if the inlet swirl centre is clocked towards the passage ($\theta^{sw} \approx 0$ and $\theta^{sw} \approx 1$). Pressure gradients at clocking positions $\theta^{sw} < 0$ and $\theta^{sw} > 1$ are not due to the actual inlet swirl core but result from the flow entrainment that takes places between two swirl cores as can be seen in Fig. 5.1. The effect of this radial pressure gradient on the migration of the swirl core and secondary flows at different clocking positions is now evaluated by analysing circumferentially mass flow-averaged NGV exit total pressure profiles. These provide information about the radial position of loss generation in the NGV passage. The profiles shown in Fig. 5.7 are derived from the same simulations shown in Fig. 5.6 with $S = 0.2$ and inlet swirl rotating in both positive and negative direction. In order to highlight the loss production due to inlet swirl, the profiles are compared to the reference case with axial inflow. Those regions with $p_t < p_{t,ref}$, i.e., higher losses, are marked red in Fig. 5.7, others blue. That is, the red zones can be attributed to loss production due to inlet swirl at the respective clocking position. The 16 clocking positions shown correspond to those in Fig. 5.3 and are grouped into Passage and Leading edge clocking. The results shown in Fig. 5.7 have been obtained using the SST-$k$-$\omega$ turbulence model. A comparison with single-row computations using the SSG Reynolds Stress model yielded qualitatively similar results (Appendix D.1.2).

Loss zones in Fig. 5.7 are assigned to four categories A to D. The classification is done based on a detailed analysis of the 3D flow field. A 3D visualisation of structures B to D can be seen in Fig. 5.8. Loss region A results from the horse shoe vortex at the LE which is particularly emphasised at swirl passage clocking. Region B is a footprint of the inlet swirl core clocked to the passage. If inlet swirl is clocked towards the vane’s LE ($3/16$ pitch $\leq \theta^{sw} < 6/16$ pitch), two loss zones occur: Region C is due to the induced LE vortex structures described by Jacobi et al. [88] (cf. Fig. 2.8) which is particularly strong at the vane’s PS and region D is the footprint of the inlet swirl core that passes the vane at its SS. The structures are visualised in Fig. 5.8 of the next section. The loss from D is especially pronounced for positive swirl because

---

3 Lowering inlet swirl to $S = 0.1$ shows the same trends qualitatively but less pronounced (Appendix D.1.2).

4 Due to the circumferential averaging, loss regions in Fig. 5.7 cannot be exclusively attributed to physical phenomena A to D but they show qualitatively the same behaviour and are therefore used for visualisation of the radial migration of swirl.

5.2 Losses and Vane Secondary Flows Induced by Inlet Swirl

109
then it rotates in opposite direction as passage cross flows at the hub (cf. Fig. 2.5). Therefore, both streams form a strong shear layer close to the hub for positive swirl orientation.

Regarding the movement of these loss zones, it can be stated that regions (B) and (D) behave in accordance with the swirl-induced pressure gradients identified in Fig. 5.6: Inlet swirl in positive direction causes an upwash at the PS and a downwash at the SS. The radial migration of the induced vortex (C), however, is against the SS pressure gradient identified in Fig. 5.6 for $0 \leq \theta^{\{sw\}} < 1/2$ pitch. Its radial position must therefore be driven by a different aerodynamic mechanism.

**Figure 5.7:** Total pressure loss profiles at NGV1 exit for different clocking positions of inlet swirl for positive (top) and negative (bottom) swirl rotation. Lower total pressure loss than in the axial reference is indicated by blue, higher by red colour, caused by a horse shoe vortex (A), inlet swirl (B and D), a swirl-induced LE vortex (C).

### 5.2.4 Leading Edge Vortices Induced by Inlet Swirl

In the previous sections the mechanism establishing a radial pressure gradient in the vane passage was outlined and it was shown that the footprint of inlet swirl at NGV outlet (B and D) moves according to this pressure gradient. The swirl-induced LE
vortex \( C \), however, moves against this pressure gradient. It is driven by another mechanisms which is investigated in this section.

**Effect of Swirl Clocking on Radial Position of Leading Edge Vortices**

The flow model by Jacobi et al. [88] describes the occurrence of a secondary flow-like structure that forms due to a roll-up of flow at the stagnation line of the vane caused by the deficit in total pressure from the incoming swirl flow (cf. Fig. 2.8). A system like this can also be observed at the right vane in the conducted simulations of the RRD-HPT. Fig. 5.8 shows the structure as a purple iso-surface\(^5\) of \( \lambda_2 = 0 \) (labelled \( C \)), for different clocking positions of inlet swirl about the LE at \( S = 0.2 \) in positive direction.

![Inlet swirl \( B \) is clocked from PS to SS of the right vane inducing a vortex \( C \) at point \( H \) where hub and shroud flow collide. Moving \( B \) to the left blocks the flow \( E \) to the stagnation line at the hub causing an acceleration \( G \) and a radial movement of \( H \). Vortex \( C \) is visualised by an iso-surface of \( \lambda_2 = 0 \) (clipped to the area of interest).](image)

**Figure 5.8:** Inlet swirl \( B \) is clocked from PS to SS of the right vane inducing a vortex \( C \) at point \( H \) where hub and shroud flow collide. Moving \( B \) to the left blocks the flow \( E \) to the stagnation line at the hub causing an acceleration \( G \) and a radial movement of \( H \). Vortex \( C \) is visualised by an iso-surface of \( \lambda_2 = 0 \) (clipped to the area of interest).

As can be seen in the top row of Fig. 5.8, the radial position of the secondary flow structure depends on the inlet swirl clocking position \( \theta^{(sw)} \): As the inlet swirl core \( B \) is clocked in positive direction from the vane’s PS to its SS, the secondary

---

\(^5\) See Jeong and Hussain [91] for a definition of \( \lambda_2 \).
flow structure $\mathcal{C}$ moves radially downwards. The radial movement can be clearly seen when comparing the current axis of vortex $\mathcal{C}$ (dotted line) with the reference location at LE clocking (dashed line) from column c). The same behaviour can be observed in the footprints of the structure in heat transfer data (resulting from a non-generic traverse) in Fig. 4.11.

The dependency of the position of the two vortices $\mathcal{B}$ and $\mathcal{C}$ can be explained as follows: Similar to the horse shoe vortices at hub and shroud the flow must be substantially decelerated in order for vortex $\mathcal{C}$ to roll up at the LE. Consequently, it must emerge at a point $\mathbf{H}$ somewhere along the vane’s stagnation line. In the top row of Fig. 5.8 those streamlines that end at the stagnation line are depicted in light red and labelled $\mathcal{E}$ (the same streamlines are shown in the bottom row with different colouring). Due to the orientation of inlet swirl $\mathcal{B}$, these streamlines form a sheet that has a positive incidence at the hub and a negative incidence at the shroud and thus runs diagonally through the flow path. The clocking of inlet swirl from PS to SS does not substantially affect the incidence of streamlines $\mathcal{E}$ at hub and shroud. Those streamlines among $\mathcal{E}$ that eventually form the induced vortex $\mathcal{C}$ (as identified by the $\lambda_2$-criterion) are coloured dark red and additionally labelled $\mathcal{F}$. It can be seen that inlet swirl $\mathcal{B}$, when clocked to the vane’s PS (column a), has to “cross” the sheet $\mathcal{E}$ from right to left and thus blocks part of the area available for $\mathcal{F}$. This blockage causes an increase in momentum of the entrained flow below the swirl core (red area labelled $\mathcal{G}$ in the bottom row). Note, that all streamlines $\mathcal{E}$ end at the vane’s stagnation line and thus do not give way to structure $\mathcal{B}$. The increased momentum $\mathcal{G}$ amplifies the upwash to the right of the swirl core, hence pushing the point of coalescence $\mathbf{H}$ of the hub and shroud part of $\mathcal{E}$ radially upwards.

In contrast, if inlet swirl is clocked to the SS (column d) it does not cross sheet $\mathcal{E}$ and thus rather blocks the flow in the upper half of the passage which leads to a deceleration of entrained flow at the hub and a downward movement of point $\mathbf{H}$. The effect is assumed to be asymmetric, i.e., more influential in the lower half, due to inclination of the hub liner.

In order to verify that swirl clocking $\theta^{\text{sw}}$ is the relevant parameter determining the radial position $r^{\text{sw}}$ of point $\mathbf{H}$ its correlations with the input parameters $X = [S, D^{\text{sw}}, r^{\text{sw}}, \theta^{\text{sw}}]$ are evaluated in the data set of the second Monte Carlo run of Tab. 5.2. The radial position $r^{\text{sw}}$ is identified by exporting a plane from the CFD flow field that is approximately aligned with the stagnation line and identifying the core of vortex $\mathcal{C}$ in this plane using the vortex identification algorithm by Graftieaux et al. [57]. The induced LE vortex is identified in 34 of the 160 simulations at swirl numbers $S > 0.15$ only. Correlation coefficients are shown in Fig. 5.9.

It can be seen that, of these four parameters, only $\theta^{\text{sw}}$ is significantly correlated
with the radial position $r_{sw}$ confirming the previous qualitative findings\(^6\). The correlation is negative, i.e., a clockwise movement of inlet swirl produces a downward radial movement of vortex which is in agreement with Fig. 5.8.

**Figure 5.9:** Correlation of inlet swirl parameters with the radial position of the induced LE vortex $r_{sw}$ at positive rotation direction. Correlations are derived from a set of $N = 34$ samples.

In summary, it can be stated that, in the simulations evaluated, the radial position of structure C in Fig. 5.7 is not governed by the radial pressure gradient induced by inlet swirl but rather by the radial position $r_{sw}$ of its formation (point H) in Fig. 5.8. A similar influence of $\theta_{sw}$ on the radial position of other passage secondary flows on the SS is visible in the 3D CFD results but these structures cannot be reliably identified by the used vortex identification algorithm. Note, that the results are based on the assumption of a steady inflow condition and must therefore be interpreted as a time-mean of a flow field that is strongly fluctuating in reality.

---

\(^6\) An influence of erroneous modelling of the inlet total pressure profile (cf. Section 4.6) has been precluded by running additional simulations with $p_{t,in} = \text{const.}$ which show the same correlation between $\theta_{sw}$ and $r_{sw}$. 

### 5.2 Losses and Vane Secondary Flows Induced by Inlet Swirl

113
5.3 Uncertainty Propagation in the Large Scale Turbine Rig

5.3.1 Description of Flow Field with Leading Edge Swirl

It is now investigated if the mechanisms identified with the generic set-up of the RRD-HPT in the previous sections affect the rig test case of the LSTR. Therefore, the uncertainty propagation workflow, illustrated in Fig. 4.13, is applied to the set-up of the LSTR NGV1 row, analysed in Section 3.4.

Before variations are imposed to the swirling inlet conditions, the datum flow field is described in the next paragraph summarising the findings by Werschnik et al. [183, 181, 182].

![Image of flow field with labels](image)

**Figure 5.10:** Secondary flows in the LSTR NGV1 row as iso-surface of $\lambda_2 = 0$ (clipped at an axial location shortly downstream of the RIDN cooling holes) for axial and swirling inflow (top), comparison of NGV pressure loss coefficient $\zeta_{NGV}$ and visualisation of streamlines going through the main loss core in ME02 (bottom)

With axial inflow at RIDN MFR = 3, a large part of RIDN coolant is carried across the passage to the PS/hub corner by its large momentum where it accumulates (Fig. 3.7), forms a coherent structure (visualised in Fig. 5.10, top, labelled J) and exits the passage at ME02. Part of the RIDN coolant, ejected directly in front of the vane, wraps up at the LE forming a second vortex (I in Fig. 5.10) that is traceable as a loss core at 30% span\(^7\) in ME02. A corresponding, weaker structure K can be observed close to the casing. Note, that none of the structures J to K in Fig. 5.10 can be found in the classical secondary flow model (Fig. 2.5 b)\(^8\).

---

\(^7\) The radial position of this washed-up vortex was found to depend on RIDN MFR in experiments.

\(^8\) The sense of rotation of vortex I opposes that of the passage vortex and those of J and K oppose the sense of the horse shoe vortices at hub and shroud, respectively. Horse show vortices at hub and shroud can be identified in CFD but are much smaller than the structures J to K.
With the swirling inflow condition shown in Fig. 4.8, clocked to the leading edge (SwL) of the left vane, the formation of structure \( \Box \) can still be observed. On the left vane it is washed-up compared to the case with axial inflow. The upwash takes place against the swirl-induced pressure gradient (cf. Fig. 5.6). It must therefore be caused directly by inlet swirl similar to structure \( \Box \) in the RRD-HPT (Fig. 5.8).

Structure \( \K \) is weakened on the left vane and a new, swirl-induced vortex \( \M \) emerges with the same sense of rotation as \( \K \). The mechanism of formation of vortex \( \M \) is different to that of vortex \( \Box \) in the RRD-HPT (Fig. 5.8) as it emerges due to a roll-up of high momentum flow impinging on the vane’s PS.

Vortex \( \Box \) at the LE cannot be observed at the swirl number \( S \) and clocking position \( \theta \{\text{sw}\} \) of the SwL configuration in Fig. 5.10. However, the structure does emerge, simultaneously with vortex \( \M \), if swirl strength is raised to \( S = 0.4 \) and swirl is clocked between 2° and 4° off the SwL conditions\(^9\). As shown in Fig. 5.11, the induced flow structure \( \Box \) changes its radial position with inlet swirl clocking position \( \theta \{\text{sw}\} \) in accordance with the findings of the previous section. An addition to these findings is that the downward movement of \( \Box \) does not depend on the direction of vane deflection which is in opposite direction in the LSTR compared to the RRD-HPT vane geometry.

Swirl clocked to the left vane leaves the secondary flows at the right vane almost unaffected, i.e., they resemble the axial configuration. The inlet swirl core \( \Box \) is deflected to the SS of the left vane and hence not visible in Fig. 5.10. It is convected to the upper half of the passage where it impinges on ME02 at 80% to 90% span.

A comparison with experiments (Werschnik [180]) of the distribution of the pressure loss coefficient \( \zeta_{\text{NGV}} \) in ME02 is shown in Fig. 5.10 (bottom). A qualitative agreement of flow features can be recognised, although the flow is less mixed out and the dominant loss core is more pronounced in CFD. A wake region behind the seven TE slots of the right vane as well as a footprint of RIDN coolant, washed-up at the right vane (RIDN \( \Box \)) to 30% span, can be recognised as described by Werschnik [180]. The influence of inlet swirl on structure \( \Box \) at

\(^9\) The narrow range of clocking positions at which vortex \( \Box \) occurs is assumed to be caused by the comparably small leading edge diameter of the vanes.
the left vane transports RIDN flow higher into the left passage than in the right one. As shown on the bottom right of Fig. 5.10, the dominant loss core in ME02 is caused not only by inlet swirl washing up on the left vane's SS but to a large part by structure $\overline{\text{M}}$ in Fig. 5.10 which rolls up in the PS/shroud corner.

5.3.2 Propagation of Uncertainty of the Inlet Traverse

Deriving Probability Densities of Uncertain Inputs

It is now investigated, how uncertainties in the swirling inlet conditions to the turbine affect the passage flow field, using the model described in Chapter 4. The inlet conditions, matched by the model, reproduce all qualitative flow features in perfect agreement as can be seen in the top right of Fig. 5.10. In order to conduct a UQ study, representative uncertainties of inputs $X$, describing the swirl field in ME01, are required because unrealistically large variations of a single parameter $X$ may overshadow others and yield to an overestimation of the variance of the outputs $Y$. All parameters $X$ are assumed to be normally distributed with mean $\mu$ and a standard deviation $\sigma$ according to Eqn. (3.19). The mean values of $X$ are known from the matching process but standard deviations must be estimated. These are derived by comparing different numerical simulations of the combustor simulator module. In addition to the reference SAS by Hilgert et al. [73] a second transient CFD of the combustor module is conducted by means of the RSM-SSG turbulence model. Information on the domain, boundary conditions, numerical mesh and set-up can be found in Appendix D.2. The mass averaged flow angle distribution at the combustor turbine interface (ME01), predicted by the SAS and SSG calculations, is shown in Fig. 1.5. The difference in the prediction of swirl angles\(^{10}\) between the different turbulence closures amounts to approximately 10°, which corresponds with a difference in swirl number $S$ of 0.05. This interval is assumed to correspond to an uncertainty of $2\sigma$. Similarly, a standard deviation in mean turbulence intensity $\overline{T_u}$ of 2.5% is derived. Deviations of swirl position, $\theta^{(\text{sw})}$ and $r^{(\text{sw})}$, are estimated from the transient orbiting of the swirl core. Therefore, the swirl centre is detected with the vortex identification model by Graftieaux et al. [57] in each time step of the transient SSG simulation. The standard deviations of its radial and tangential position are used as representative inputs to the UQ analysis.

Results

The standard deviations for the input parameters $X$ used in this study, are summarised on the left of Tab. 5.2. Two UQ studies a) and b) are conducted with $n = 2$

\(^{10}\) The differences in flow angles in the RSM simulation are mainly due to errors in the prediction of axial velocity in ME01. The agreement in circumferential velocity $u_{\text{tan}}$ is better than that of the flow angles (not shown).
and \( n = 4 \) uncertain parameters, respectively. Within the investigated parameter range, no substantial change in flow topology through the passage occurs, i.e., the inlet swirl core is convected to the vane SS, as described in the previous section, for all sampled inlet conditions. The effect on overall pressure loss coefficient \( \zeta_{\text{NGV}} \) between ME01 and ME02 is shown in terms of a standard deviation and correlations of the individual inputs \( X \) with \( \zeta_{\text{NGV}} \) in Tab. 5.2. Correlations \( \rho \) are computed from the Monte Carlo samples \( \tilde{Y} \).

Table 5.2: Uncertainty Propagation through NGV1 of the Large Scale Turbine Rig

| Standard deviations \( \sigma \) of \( X \) | \( \mu \pm \sigma \) of \( Y \) | Correlation coefficients \( \rho \) |
| \( \theta^{(\text{sw})} \) (\(^\circ\)) | \( r^{(\text{sw})} \) (%) | \( S \) (-) | \( \overline{T}_u \) (%) | \( \zeta_{\text{NGV}} \) (-) | \( \rho(\theta^{(\text{sw})}, \zeta_{\text{NGV}}) \) | \( \rho(r^{(\text{sw})}, \zeta_{\text{NGV}}) \) | \( \rho(S, \zeta_{\text{NGV}}) \) | \( \rho(\overline{T}_u, \zeta_{\text{NGV}}) \) |
| a) | 1.5 | -0.025 | 2.931±0.048 | -0.76* | - | 0.84* | - |
| b) | 1.5 | 2.5 | 0.025 | 2.964±0.068 | -0.09 | -0.30 | 0.71* | 0.71* |

\( \dagger \) \% of span height; The superscript * indicates significant correlations.

The swirl number \( S \) has a significant positive influence on \( \zeta_{\text{NGV}} \) which is in agreement with expectations due to the increased blade loading and pressure loss at higher \( S \). A negative correlation with swirl clocking position \( \theta^{(\text{sw})} \) in run a) can be recognised, i.e., clocking inlet swirl to the left, away from the LE, increases pressure loss. This effect is overshadowed by the effect of inlet turbulence \( \overline{T}_u \) in run b). The correlation of \( \overline{T}_u \) with \( \zeta_{\text{NGV}} \) is also positive which indicates increasing pressure loss from increased inlet turbulence in agreement with previous studies (Schmid et al. [159]). The radial position of inlet swirl \( r^{(\text{sw})} \) does not have a significant effect within the investigated uncertainty band. Effects on film cooling and heat transfer on the NGV hub end wall are shown in Fig. 5.12.

The uncertainty in \( \eta_{\text{FC}} \) is mostly below 5 \% and distributed similarly in both passages with highest values close to the vane PS whereas the uncertainty in Nu is larger left of the vane impinged by swirl. Note, that the epistemic uncertainty of \( \eta_{\text{FC}} \), due to the modelling of turbulent diffusion by means of the turbulent Prandtl number, is much higher than that due to swirl inlet conditions (Fig. 3.15).

Correlation coefficients show that increasing \( S \) disturbs the cooling film and generally decreases \( \eta_{\text{FC}} \) on the hub. The area around the right vane, where \( \eta_{\text{FC}} \) is uncorrelated with \( S \), is assumed to occur due to an overshadowing by the effects of inlet swirl clocking \( \theta^{(\text{sw})} \) which increases film cooling effectiveness in this region.

The correlation of \( \theta^{(\text{sw})} \) with \( \eta_{\text{FC}} \) is in opposite directions at the PS/hub corners of the left and right vane, respectively. The opposing trends at these locations (where highest deviations of \( \eta_{\text{FC}} \) occur) are in line with an opposing radial migration of PS secondary flows as discussed below (downwash on the left vane, upwash on the right vane).

5.3 Uncertainty Propagation in the Large Scale Turbine Rig
A similar trend in the influence of \( \theta^{\{sw\}} \) on \( \text{Nu} \) can be identified only in the rear part of the hub. This indicates that the deviations of \( \eta_{\text{FC}} \) at the PS/hub corners are rather caused by changes in the radial transport of coolant from the vane than a disturbance of the RIDN film. Increasing inlet turbulence intensity \( \overline{T}u \) generally decreases \( \eta_{\text{FC}} \) but increase \( \text{Nu} \) by an increase in turbulent diffusion.

**Mean and Standard Deviation**

![Mean and Standard Deviation](image1)

**Correlation Coefficients**

![Correlation Coefficients](image2)

**Figure 5.12:** Mean \( \mu \) and standard deviation \( \sigma \) of film cooling effectiveness and Nusselt number at the hub end wall (top, computed from the gPC expansion \( \tilde{Y} \)) and correlations \( \rho \) with the parameters determining the inlet traverse (bottom, computed from the Monte Carlo samples \( \tilde{Y} \)).

Effects of swirl uncertainty on film cooling and heat transfer of the vanes are shown in Fig. 5.13. These quantities are not investigated experimentally but analysed here to illustrate the deviations of passage secondary flows.

The mean values of \( \eta_{\text{FC}} \) and \( \text{Nu} \) reflect the observations about the secondary flows in Fig. 5.10. Fluid from the RIDN is washed up on the vane PS by structure \( \Box \) and, on the left vane, flow from the casing is additionally washed down at the PS by...
structure $[\mathbb{M}]$. On the right vane’s PS, on the contrary, the effect of vortex $[\mathbb{K}]$ is visible as increased cooling effectiveness and heat transfer along a streak between 80 and 90% span height. The influence of residual inlet swirl $[\mathbb{B}]$ can be observed on the SS of the left vane where coolant is washed up towards the casing. Considering the deviations $\sigma(\eta_{FC})$, it can be seen that structures $[\mathbb{I}]$ and $[\mathbb{M}]$ are mainly responsible for the variations in $\eta_{FC}$ and Nu.

### Mean and Standard Deviation

#### Film Cooling Effectiveness

- **Left Vane**
- **Right Vane**

#### Heat Transfer

- **Left Vane**
- **Right Vane**

### Correlation Coefficients

#### Film Cooling Effectiveness

- $\rho(S, \eta_{FC})$
- $\rho(\Delta T^{(sw)}, \eta_{FC})$
- $\rho(\Delta T^{(sw)}, \eta_{FC})$
- $\rho(\Delta T_{U}, \eta_{FC})$

#### Heat Transfer

- $\rho(S, \eta_{FC})$
- $\rho(\Delta T^{(sw)}, \eta_{FC})$
- $\rho(\Delta T^{(sw)}, \eta_{FC})$
- $\rho(\Delta T_{U}, \eta_{FC})$

**Figure 5.13:** Mean $\mu$ and standard deviation $\sigma$ of film cooling effectiveness and Nusselt number on the SS ($s < 0$) and PS ($s \geq 0$) of the vanes (top, computed from the gPC expansions $\tilde{Y}$.) and correlations $\rho$ with the parameters determining the inlet traverse (bottom, computed from the Monte Carlo samples $\tilde{Y}$.). Correlations are shown quantitatively in regions of high deviations only ($\sigma(\eta_{FC}) \geq 1.5\%$ and $\sigma(Nu) \geq 300$) for better visibility. Labels correspond to the structures visualised in Fig. 5.10.

The correlation coefficients in Fig. 5.13 reveal the source of these deviations which are caused on both vanes almost exclusively by swirl clocking $\theta^{(sw)}$. The swirl rotates in counter-clockwise direction (cf. Figure 4.8) and $\theta^{(sw)} > 0$ moves swirl
in clockwise direction, towards PS of the left vane. According to the findings of Section 5.2, this corresponds to a downward movement of the flow along the PS of this vane which is confirmed by the correlations with film cooling effectiveness. As $\rho(\theta^{sw}, \eta_{FC}) > 0$ below the cold streak $\Theta$ and $\rho(\theta^{sw}, \eta_{FC}) < 0$ above it, a downward movement of the cold streak $\Theta$ occurs for positive clocking of swirl (indicated by a black arrow in Fig. 5.13).

The situation is reversed at the right vane which is not exposed to inlet swirl. Here, the streak $\Theta$ on the PS moves upwards if $\theta^{sw} > 0$. Increasing the swirl number $S$ supports the trends of $\theta^{sw} > 0$ on the left vane. Effects of radial swirl position $r^{sw} > 0$ and mean inlet turbulence intensity $Tu$ are not visible\textsuperscript{11} in the regions of interest. Correlations with Nusselt number are shown in Appendix D.4. The findings support the explanation of the effect of $\theta^{sw}$ on $\eta_{FC}$ in Fig. 5.12.

**Influence of Inlet Total Pressure Profile**

As pointed out in Section 4.6, there is an unavoidable source of error in calculating the inlet total pressure, because the “correct” distributions of axial velocity $u_{ax}(r, \theta)$ and pressure $p(r, \theta)$ for a given sample of $X$ are unknown. In order to estimate how the distribution of $p_t$ (and therefore $u_{ax}$ and $p$) influences the qualitative results of this chapter, run b) is repeated using the fixed datum pressure profile $p_t^{\text{ref}}$ for all $N$ simulations, only varying inlet flow angles and turbulence. It can be seen in Fig. 5.14 that the deviations, resulting from this approach (right), are qualitatively equal to those in Fig. 5.13 (agreement with other results is similar, see Appendix D.3). This indicates that the influence of a false calculation of total pressure on the qualitative findings of this section is negligible.

\textsuperscript{11} Note, that correlation coefficients $\rho$ close to zero do not imply that these parameters $X$ have no influence on the evaluated quantities $Y$ at all but rather that their influence (if present) is smaller than that of the other uncertain parameters $X$. 

---

5 Results and Discussion
5.4 Uncertainty Propagation in the Engine 3E

5.4.1 Geometry and Numerical Set-up

In addition to the UQ analysis of the LSTR a realistic HPT configuration with a non-homogeneous inlet temperature profile is now investigated. Simulations are conducted on the Engine 3E\(^{12}\) (E3E), a two-shaft demonstrator core engine in the medium thrust range developed by RRD, as described by Klinger et al. [97]. The engine is equipped with a low-NO\(_x\), lean burn combustor in single annular arrangement. The two-stage, shroudless HPT has contoured end walls at both NGV1 hub and shroud and the rotor hub in order to suppress the formation of secondary flows. The NGV1 hub is cooled from a continuous RIDN slot. Ballistic cooling applied in the engine, as described by Benton et al. [17], is not considered in this work. Both, NGV1 and Rotor1, are film-cooled, the cooling of NGV2 is internal.

![Combustor and Turbine Geometry](image)

![3D Model and Distribution of y⁺](image)

**Figure 5.15:** Geometry of E3E combustion chamber and high pressure turbine section (top, adapted from Klinger et al. [97]). Numerical set-up for simulations of 1.5 stages of the turbine (bottom left) and distribution of y⁺ (bottom right)

\(^{12}\) 3E is an acronym for “environmental friendliness, efficiency and economy”.
The time-averaged 2D combustor exit condition obtained by means of combustor CFD at 20% pilot flame is provided by RRD as reported by Schmid [157]. Since no distributed values at inlet are available for two-equation turbulence closure, turbulence intensity and length scale are assumed constant with $T_u = 20\%$ and $\mathcal{L} = 0.02\,\text{m}$, respectively. The swirl number at the domain inlet is $S = 0.16$. The simulated combustor domain is shown by Pyliouras [136] and Lazik et al. [110].

The engine contains 14 burners and 20 NGVs. In agreement with Schmid [157], the NGV count is scaled to 21 in order to allow for a numerically efficient set-up of a $51.43^\circ$ sector with three NGVs and two burners. With the given boundary conditions, the turbine is in cruise condition and the first nozzle's throat is choked. The numerical model is shown in Fig. 5.15. One and a half of the two HPT stages are modelled. NGV and rotor inner cavities are not included in the model in order to improve mesh quality and to increase numerical stability and efficiency. Instead, the domain is cut through the cooling holes and each hole is assigned an individual inlet to the numerical domain. The holes are cut-off at lengths of approximately 3.5 to 5 hole diameters. The integral design mass flow of each row of cooling holes is specified as boundary condition, together with the design coolant total temperature. The consideration of film cooling requires an unstructured meshing of the NGV1 and Rotor1 rows (using Centaur 10.6 [171]), a structured mesh is used to discretise the internally cooled NGV2 row. The meshes resolve the boundary layer with a non-dimensional wall distance $y^+$ below 3 in most parts of the domain. High values of $y^+$ occur at the NGV1 throat, the NGV2 seal cavity and within the coolant holes. An overview of grid size and quality is given in Tab. 5.3.

| Table 5.3: Numerical Set-up for Simulations of the Engine 3E |
|----------------|----------------|----------------|-------------------------|
| Numerical Mesh | Set-up and Convergence |
| Domain Passages | NGV1 Cells (Mio.) 31.5 | Rotor1 Cells 13.5 | NGV2 Cells 3.5 |
| | Max. $y^+$ 17.7 | Avg. $y^+$ 0.87 | Max. EVR $^1$ 556.4 |
| | Avg. $y^+$ 0.70 | Avg. EVR 2.4 | 13.7 |
| | Min. cell angles ($^\circ$) 14.0 | 11.7 | 27.0 |
| Solver | ANSYS® CFX® v17.0, steady |
| Gas model | ideal, compressible |
| $c_p, k = f(T)$ | SST-k-ω, fully turbulent |
| Turbulence | Discretisation $^\dagger$ $\bar{\beta} \geq 0.84$ (for $u, P, h$) |
| $\bar{\beta} \geq 0.60$ (for $k, \omega$) | Residuals $< 1 \times 10^{-4}$ |
| Imbalances $< 0.02\%$ |

$^1$ Element volume ratio $^\dagger$ Flux-blending as in Eqn. (3.13), averaged over entire domain, smallest average is reported
The mesh is refined in the wake regions behind the nozzles and rotor blades by a factor of 0.5 in cell size. The SST-\(k-\omega\) model is used for turbulent closure at a turbulent Prandtl number of \(\text{Pr}_t = 0.9\). Grid dependency of the evaluated integral quantities is investigated by conducting a grid refinement study, in which the NGV1 and Rotor1 mesh are refined between 16.9 and 124.3 Mio cells total. The final mesh at 49.1 Mio cells is derived from the second coarsest mesh by local refinements. Single row simulations of NGV1 are conducted using the SSG Reynolds stress model in order to evaluate the influence of turbulence modelling on the results. A circumferentially averaged pressure profile from the coupled simulation is used as outlet condition.

The results shown in Fig. 5.16 indicate that stage efficiency \(\eta\) and NGV pressure loss...
$\zeta_{\text{NGV}}$ are not fully grid independent with the final mesh (49.1 Mio cells). However, as shown in Tab. D.3, this discretisation error mainly causes an offset in the expected value of the uncertain outputs $Y$ (not their variance) and can therefore be regarded acceptable for uncertainty propagation analyses. The differences in predicted NGV1 wall temperatures between the 49.1 and the 124.3 Mio cell mesh, shown in Fig. 5.16, are also regarded small for the purpose of this study. The histograms in the centre of the figure illustrate the fractions of NGV surface area $\delta A/A$ with a certain wall temperature $T_w$ for the different grids. The distributions are mostly converged on the 49.1 Mio cell mesh. The effect of turbulence modelling on wall temperatures is shown in Appendix D.5.2.

5.4.2 Description of Flow Field with Swirling Inflow

Before the actual uncertainty propagation of the inlet profile is conducted, the baseline configuration with the 2D inhomogeneous lean burn inlet BC$^{13}$ (Fig. 1.2b) is examined. The flow field in the HPT nozzle is shown in Fig. 5.17 next to a reference obtained with axial inflow conditions and a homogeneous temperature distribution. With axial inflow (right) the cooling film is undisturbed and distributed evenly on the surface of the hub as well as on the vanes. The stagnation line at the LE can be identified, separating the cooling film to the suction and pressure side of the vane. The flow from two rows in the proximity of the stagnation line is not immediately transported to the vane surface but can be observed to flow radially downward along the stagnation line. The kidney vortices from these cooling holes merge into a coherent structure $\text{N}$ between 30 and 40% blade height. The structure merges with the hub horseshoe vortex $\text{O}$ and both enter the blade passage.

In case of inlet swirl, secondary flows are distributed differently among the three vanes due to the different relative clocking positions of swirl cores and vanes (Fig. 5.18 top). All three vanes show structures similar to $\text{N}$ and $\text{O}$ from the reference, yet less pronounced. Additionally, a large horseshoe vortex $\text{A}$ at the shroud can be identified. This vortex is not present in the axial case which can be explained by the local thickness of the incoming boundary layers in front of the vanes: Both cases, axial and with inlet swirl, have the same mass flow averaged level of total pressure at inlet. With swirling inflow, however, $p_t$ is distributed inhomogeneously. At the casing $p_t$ is lower than the homogeneous pressure level of the reference configuration due to the swirl-induced increase in dynamic pressure outside the boundary layer (Fig. 5.18 bottom, at 20 and 80% span). As the distribution of static pressure at the casing is determined mainly by the potential field of the vanes, a

$^{13}$ Turbine inlet conditions predicted by combustor CFD show a relatively low total pressure close to the casing due to an interaction with RODN inflow. In this work, the prescribed inlet $p_t$ is locally increased at $h_{\text{rel}} > 90\%$, compared to the inlet conditions used by Schmid [157], in order to prevent backflow at the domain inlet.
low velocity results upstream of the stagnation line. Due to the reduction of total pressure at the casing from inlet swirl, velocity in front of the vanes becomes very low, the associated boundary layers grow and favour the rolling-up of the horseshoe vortices at the shroud. The positive incidence at the casing and the vane geometry cause vortices to form primarily at the vane PS.

![Diagram of NGV1 secondary flows and wall temperature](*Figure 5.17: NGV1 secondary flows and wall temperature: Secondary flows are visualised by an iso-surface of $\lambda_2 = 0$. Cuts through the LE vortices of the left vane computed with different turbulence models are shown on the left, distributions of a reference case with homogeneous, axial inflow on the right.*)

The swirl-induced flow structure, described in the first part of this chapter ([C] in Fig. 5.8), can be clearly identified at the LE of the left vane. In Fig. 5.17, vortex [C] at approximately 40% span has an opposite direction of rotation as the horse shoe vortex [A]. A large, counter-rotating vortex [C'] can be identified as well. Thus, [C] and [C'] form the induced vortex pair, described by Jacobi et al. [88] (Fig. 2.8). Structure [B], the residual inlet swirl core, rotates in same direction as [C] and migrates through the passage at midspan of the PS. The central vane, which is only remotely exposed to inlet swirl, shows weak versions of [C] and [B] whereas the right vane, which is not exposed to inlet swirl, shows no such structures but qualitatively resembles the axial reference case as in the LSTR. Simulations of the NGV1 row, that are run with both, two-equation and RSM turbulence models, show, that the location and extent of the induced vortex pair is insensitive to turbulence modelling (Fig. 5.17, left). Distributions of adiabatic wall temperature on the vane in Fig. 5.17 show that high temperatures occur primarily in regions where the cooling film is disturbed (the LE stagnation line and the PS of the left vane). The distribution of the RIDN film

5.4 Uncertainty Propagation in the Engine 3E
at the hub can be seen to be circumferentially homogeneous in the reference case. It is clearly influenced by inlet swirl and causes an accumulation of cooling flow entering the passage on the SS of the central vane.

5.4.3 Matching of Inlet Conditions

A parameterised 2D inlet condition is matched to the traverse from combustor CFD as described in Section 4.10.1 and applied to the HPT set-up. Distributions of $u_\text{ax}(r, \theta)$ and $p(r, \theta)$ from the datum field are used in the matching.

![Figure 5.18: Comparison of datum and matched 2D inlet boundary conditions for temperature and flow vectors (top) and circumferentially averaged conditions at domain inlet and NGV1 outlet (bottom) from CFD. Results of a simulation with axial inflow conditions are shown as reference.](image)

Results from CFD, using these matched inlet conditions, are compared to the datum inlet BC in Fig. 5.18 and Fig. 5.19. Circumferential averages of aerodynamic data at domain inlet and behind the first nozzle are shown in Fig. 5.18 for both 2D inlet conditions and the axial reference. Compared to the CFD results obtained by Schmid [157] the hot streak radial location at NGV outlet is about 10% higher and the total pressure loss core at midspan is more pronounced which may be due to the addition of film and TE cooling in the simulations in this work. Similarly to the matching of the LSTR inlet condition (Fig. 4.8), the largest error at
NGV inlet is in the matching of the swirl flow angle $\alpha_{\text{tan}}$ between 60 and 90 % span which also leads to a mismatch of NGV1 outlet swirl. The other quantities, especially the temperature traverse, are matched with good agreement. It is acknowledged, however, that despite the good qualitative matching differences to datum operating conditions of $\Delta T_w(r, \theta) = 0.92\%$, $\Delta p(r, \theta) = -0.013\%$ and $\Delta \dot{m} = -0.45\%$ occur. In the post-processing of the UQ analysis this difference in $\bar{T}_{\text{t,in}}$ is corrected for as described in Section 5.4.4.

![Qualitative distributions of NGV1 wall temperature of the matched inlet traverse and a quantitative comparison with datum conditions](image)

**Figure 5.19**: Qualitative distributions of NGV1 wall temperature of the matched inlet traverse (top) and a quantitative comparison with datum conditions (bottom), where each bin corresponds to a temperature range of $\Delta T_w = 10 K$. Red bars indicate the averaged values of each area.

The adiabatic wall temperature maps in Fig. 5.19 indicate that the qualitative distribution of wall temperature and its characteristic features, shown in Fig. 5.17, are reproduced by the matched BCs. Largest differences can be seen at the central vane and the hub end wall of the central and right passage. Secondary flow structures (not shown) agree well with the results shown in Fig. 5.17. Differences in the position of the stagnation lines of the central and right vane can be observed which can also be seen in the distributions of wall temperature in the stagnation region of these vanes. The distribution of Nusselt number, shown in Appendix D.6, is of similar agreement as the matched wall temperatures.

### 5.4.4 Propagation of Uncertainty of the Inlet Traverse

It has been shown in the previous sections that the CFD set-up can sufficiently resolve the aero-thermal quantities of interest $Y$ in the first E3E high pressure turbine stage and the model described in Chapter 4 yields a 2D inlet BC that can represent the...
characteristic features of the datum conditions from combustor CFD. The aim of the following analysis is to quantify how uncertainties in the prediction of these 2D turbine inlet conditions propagate through the turbine and how they affect quantities $Y$.

**Deriving Probability Densities of Uncertain Inputs**

As shown in the first part of this chapter, the inlet swirl strength $S$ and its clocking position $\theta^{\text{sw}}$ are the most significant inlet swirl related parameters. In addition, the effects of hot spot temperature $T^{\text{hs}}_{\text{max}}$ and position, $\theta^{\text{hs}}$ and $r^{\text{hs}}$, are considered in the analysis. The set of uncertain input parameters is therefore given by

$$X = \left[ T^{\text{hs}}_{\text{max}}, r^{\text{hs}}, \theta^{\text{hs}}, S, \theta^{\text{sw}} \right].$$

(5.1)

The uncertainties of these parameters must be quantified by PDFs. As in Section 5.3, a normal distribution of all parameters is assumed and the respective standard deviations $\sigma$ must be estimated. The same deviations as for the LSTR are assumed for the swirl related parameters (Tab. 5.2). In order to determine the deviations of the hot spot, $T^{\text{hs}}_{\text{max}}$, $r^{\text{hs}}$ and $\theta^{\text{hs}}$, a number of sector temperature measurements along the annulus of an engine combustor are matched with the traverse model and the standard deviations of the respective parameters are computed. The measured and matched traverses are shown in Appendix D.7. Note, that the data are taken from a different low-NO$_x$ combustor than the one installed in the E3E, because no such data are available for the investigated configuration. The standard deviations for all uncertain parameters are rounded to characteristic values which can be seen in the left of Tab. 5.4. Note, that these parameters must be understood as characteristic values because they are derived from different combustor and turbine geometries.

**Uncertainty Propagation Analysis and Temperature Correction**

Uncertainties of outputs $Y$ are determined in four different gPC runs of second order with different subsets of uncertain input parameters $X$ varied (Tab. 5.4). Due to limitations in computational resources, the number of samples in each run is limited to $N \approx 20$ as shown in Tab. 5.4. The parameters describing hot spot position and temperature $T^{\text{hs}}_{1,\text{max}}$ apply for the region labelled “hot spot” in Fig. 1.2. For high $T^{\text{hs}}_{1,\text{max}}$ the maximal temperatures of the hot zone close to the casing is decreased in order to match operating conditions as described in Section 4.7.4.

The overall workflow of each UQ run is shown in Fig. 4.13. Inlet velocity$^{14}$ is

---

$^{14}$ As shown in Fig 5.15 an inlet profile spanning over two equal combustor sectors is applied to three nozzle passages. Due to the different clocking positions relative to the stator vanes, axial velocities $u_{\text{ax, left}}(r, \theta)$ and $u_{\text{ax, right}}(r, \theta)$ in the two sectors are different. In order to simplify the
corrected in two successive CFD calculations as illustrated in Fig. 4.13. The initial run is iterated for about 1.5 throughflow times (through the 1.5 stages) with a relatively large pseudo-timestep of $\Delta t = 1 \times 10^{-5}$ s. The second run is restarted from the initial run, using the corrected inlet traverse, and iterated for 3 throughflow times with a smaller pseudo-timestep of $\Delta t = 5 \times 10^{-6}$ s.

![Figure 5.20: A change in hot spot temperature is balanced by the background profile. For large $T_{3,\text{max}}^{(\text{hs})}$ the temperature $T_{2,\text{max}}^{(\text{hs})}$ of the hot zone close to the casing is additionally reduced to conserve $T_{t,\text{in}}$.](image)

As discussed in Section 3.3.3, the comparison of turbine efficiency with 2D inlet conditions can be misleading if the averaged inlet temperature to the turbine varies. As described in Section 4.7, the inlet temperature is scaled to operating conditions in the UQ simulations. This approach, however, is not yet sufficient to achieve comparable data, since the remaining variation of $\overline{T_{t,\text{in}}}$ is still at the order of 0.5% after the scaling process (due to interpolation errors, model tolerances, a finite number of iterations in the correction and differences between the two inlet sectors). These deviations are corrected for by the following procedure: The relation of output quantities $Y$ with inlet temperature $T_{t,\text{in}}$ is assumed linear in the considered regime and corrected by

$$Y^* = Y + \Delta T_{t,\text{in}} \frac{\Delta Y}{\Delta T_{t,\text{in}}},$$

(5.2)

where $Y^*$ represents the corrected quantities. The temperature sensitivities $\Delta Y/\Delta T_{t,\text{in}}$ are obtained by running multiple simulations with varying levels of a homogeneous inlet temperature (Fig. 5.21) and fitting linear regression curves to the data. The plots show that the assumption of a linear dependency of the considered quantities with temperature is justified in the relevant range. The trend of negative efficiency changes with $T_{t,\text{in}}$ is in agreement with theoretical considerations and the findings of Schmid [157]. Changes in turbine power $P$ with averaged inlet temperature are very small (less than 0.5 per mill) which indicates that the

... correction process, axial velocity is not evaluated individually for both sectors but their average $\overline{u_{\text{ax}}(r, \theta)} = (u_{\text{ax,left}}(r, \theta) + u_{\text{ax,right}}(r, \theta))/2$ is used in the model.

Note, that, even if only parameters are varied that determine the velocity distribution (gPC runs a and b in Tab. 5.4), changes in the temperature background field are made to maintain operating conditions.

5.4 Uncertainty Propagation in the Engine 3E
efficiency loss with temperature in Fig. 5.21 is mainly due to the influence of the inlet temperature level. The temperature correction is applied to all quantities reported in the following, except for $T_{t,in}$ and polytropic efficiency $\eta'$.

![Figure 5.21: Inlet temperature correction of 0D results. Three simulations are run with homogeneous inlet temperature distribution at different temperature levels. Stage parameters are evaluated for each run and a linear regression (dashed line) is fitted to the data.](image)

5.4.5 Results

Stage Efficiency and Capacity

Uncertainty propagation with respect to isentropic efficiency of the first stage $\eta$, Eqn. (2.7), polytropic efficiency $\eta'$, Eqn. (2.8), and NGV1 pressure loss $\zeta_{NGV}$, Eqn. (2.9), is shown in Tab. 5.4. Four gPC runs a) to d) are analysed, each with different subsets of varied parameters $X$ (Eqn. 5.1). The data shown are computed from gPC expansions $\tilde{Y}$ of the random variables. The influence of temperature correction, statistical post-processing and mesh resolution on the results is shown in Appendix D.8.1.

Histograms of deviations of $\eta$ and $\eta'$ from the reference case are shown below the table. It can be seen that the deviations are in qualitative agreement, i.e., the effect of the temperature ratio on $\eta$ is eliminated by the temperature correction, Eqn. (5.2). The temperature scaling (Fig. 5.20) and iterative correction steps of the process (cf. Fig. 4.13) are expected to introduce systematic offsets between the mean values of efficiency $\mu(\eta)$ and $\mu(\eta')$ and the reference which can be seen in the histograms. As these offsets cannot be corrected for a posteriori, they are ignored and only deviations $\sigma$ from the mean are included in the discussion.

The most realistic estimate of uncertainty in isentropic efficiency is obtained from run d) where temperature and inlet swirl are varied at the same time. For this case $\sigma(\eta) = 0.17\%$. The separate variations of inlet swirl (run a) and the hot spot (run c) each show lower deviations in $\eta$ which indicates a contribution of both to the overall uncertainty in run d).

In Section 5.2.2, a dependency of $\eta$ on inlet swirl number $S$ is identified. Comparing runs a) and b) shows that doubling the uncertainty in $S$, at an equal uncertainty
of $\Delta \theta^{(\text{sw})}$, almost doubles the uncertainty of $\eta$ (increase by a factor of 1.8). This rise in $\sigma(\eta)$ is associated to a rise in $\sigma(\zeta_{\text{NGV}})$ by the same factor and is therefore assumed to occur primarily due to NGV pressure loss in agreement with the findings of the previous chapters.

Table 5.4: Uncertainty Propagation through the Engine 3E HPT

<table>
<thead>
<tr>
<th>Standard deviations of inputs $X$</th>
<th>Mean ± standard deviation of outputs $Y$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T^{(\text{hs})}_{\text{max}}$ (%)</td>
<td>$\eta$ (%)</td>
</tr>
<tr>
<td>$r^{(\text{hs})}$ (%)^†</td>
<td>$\theta^{(\text{bs})}$ (%)</td>
</tr>
<tr>
<td>a)</td>
<td>-</td>
</tr>
<tr>
<td>b)</td>
<td>-</td>
</tr>
<tr>
<td>c)</td>
<td>2.5</td>
</tr>
<tr>
<td>d)</td>
<td>2.5</td>
</tr>
</tbody>
</table>

* % of $T_{\text{in}}$; † % of span height; ^* % of datum conditions;

Deviations in hot spot position and temperature (at a fixed inlet swirl field), run c), cause deviations in $\eta$ which are larger than those induced by deviations in swirl (at a fixed temperature field) in run a). The deviation in $\zeta_{\text{NGV}}$, however, is much smaller in case c) which indicates that efficiency deviations due to uncertainty of the swirl field are primarily caused by NGV pressure loss whereas those due to temperature deviations are rather caused by losses in the rotor.

Note, that run c) is the only one which introduces a notable skew to the distribution of $\eta$ (possibly by the temperature scaling, Fig. 5.20). This is, however, mitigated by taking variations of swirl into consideration (run d).

The influence of the different parameters in $X$ on results $Y$ is quantified by correlation matrices$^{16}$ shown in Fig. 5.22. If swirl parameters are varied only (run a, left), swirl strength $S$ is correlated strongly negatively with stage efficiency $\eta$ (red box) which can be explained by the positive correlation with viscous pressure loss in the NGV $\zeta_{\text{NGV}}$ (blue box) in agreement with Fig. 5.2. If hot spot parameters are varied only (run c, centre), a strong positive correlation of radial hot spot position $r^{(\text{hs})}$ with

$\Delta \eta$ (%) 

$\Delta \eta'$ (%) 

---

16 The coefficients $\rho(X, Y)$ are computed from a separate sampling of the expansions $\tilde{Y}$ in Eqn. (3.28) with $N = 100,000$ points in order to grant statistical significance.
η is evident. Both, an increase in power as well as a decrease in losses ζ_{s,R} in the rotor, contribute to the increase in η. The physical background to these correlations are investigated below. The trends in the weak correlations of swirl clocking θ^{sw} and hot spot temperature T_{t,max}^{hs} with η in runs a) and c) can be explained similarly by changes in power.

If both, swirl and the hot spot, are allowed to vary at the same time (run d, right), their effects overshadow each other: NGV pressure loss (blue) is determined by the swirl number whereas stage efficiency (red) is determined by the hot spot. This observation further confirms the thesis that swirl is primarily important for losses in the nozzle whereas the hot spot determines rotor losses (Beard et al. [15]). Although the trends in these correlations are the same as in the separate investigations a) and c), the results of run d) emphasise the necessity to investigate the quantitative effects of the aerodynamic and thermal fields on efficiency in a combined approach. Clocking effects, θ^{sw} and θ^{hs}, have almost no impact in run d). Capacity Γ is determined by swirl number S which increases aerodynamic blockage in the passage and decreases \dot{m}_{in}.

Two aspects must be kept in mind when interpreting the results in Fig. 5.22: First, the effects of swirl and hot spot clocking on the rotor are assumed to be strongly blurred by the mixing plane downstream of the NGV row. Also, one would expect non-linear correlations of these parameters with efficiency (as in Fig. 5.3) which cannot be detected by linear statistics. Correlation coefficients for these input parameters are therefore assumed not to be meaningful.

Secondly, the given correlations are valid only for the assumed distribution of input parameters X in Tab. 5.4. If the variance in the “real” process differs strongly from that assumed for this study, the relative influence of hot spot and swirl variations...
may be different.
The physical relation of the radial position of the hot spot \( r^{\text{hs}} \) with stage efficiency is now investigated by analysing the statistics of the simulations of run d). The data in Fig. 5.22 indicate that the increase in efficiency with \( r^{\text{hs}} \) is caused by an increase in power\(^{17}\) rather than a decrease of losses \( \Delta s \) (\( r^{\text{hs}} \) is not correlated with the loss coefficients \( \zeta \)). According to the Euler turbine equation (2.5), the influence of \( r^{\text{hs}} \) on power must take place via the circumferential flow component at NGV exit, \( \Delta C_u \), which are positively correlated.

The effect of the radial hot spot location on \( C_u \) at NGV exit can be explained by preferential heating (Section 2.4.1) which is analysed by regarding the radial profiles of temperature \( T_t(r) \) and rotor relative incidence \( \Delta \beta_{\tan}(r) \) at NGV1 exit and specific entropy \( s \) behind the rotor that are shown in Fig. 5.23 for gPC run d). The curves at NGV1 exit display minimal deviations (grey bands) at 40\% span which increase towards the end walls. These local deviations of temperature and incidence are strongly correlated at each radial position (\( \rho(T_t, \Delta \beta_{\tan}) = 0.72^* \), on spanwise average) which confirms the occurrence of preferential heating: Locally high temperatures increase circumferential velocity \( C_u \) and thus also (positive) incidence \( \Delta \beta_{\tan} \).

The blue to red colouring according to the local correlation with \( r^{\text{hs}} \) indicates that both, temperature and incidence, are increased towards the casing and decreased towards the hub by \( r^{\text{hs}} \) because moving the hot spot towards the casing increases the outlet temperature above 40\% span and decreases it below. The increase in overall power with \( r^{\text{hs}} \) can be explained by the larger area of the annulus with higher radius that increases the weight of \( \Delta C_u \) in averaging.

Figure 5.23: Circumferential averages and variance of total temperature \( T_t \) and rotor relative incidence \( \Delta \beta_{\tan} \) (positive for \( \Delta \beta_{\tan} > 0 \)) at NGV1 outlet and normalised entropy \( s \) at rotor outlet (left) next to rotor blade loading at 90\% span (right). Colours indicate correlation with hot spot radial position \( r^{\text{hs}} \). Note, that variances of 3 and 5 \( \sigma \) are shown for clear visualisation.

\(^{17}\) Next to power \( P \), a positive correlation of \( \Delta h_t \) with \( r^{\text{hs}} \) can be identified (\( \rho(r^{\text{hs}}, \Delta h_t) = 0.73^* \)) which rules out a possible bias by changes in \( m_{\text{in}} \).
The deviations of incidence $\Delta \beta_{\text{tan}}$ influence entropy generation $\Delta s$ in the rotor by increased blade loading at the front-loaded rotor tip shown on the right in Fig. 5.23. The variations in $\Delta \beta_{\text{tan}}$ cause a variation of pressure primarily in the front part of the SS. The blue colouring indicates a negative correlation with the radial hot spot position $r^{(\text{hs})}$ at that location. That is, a hot spot moving towards the casing decreases pressure at the SS, hence aerodynamically loads the blade at the tip and increases viscous losses which explain the rise of $s$ at rotor outlet. Although a clear correlation of $r^{(\text{hs})}$ with local entropy behind the rotor can be identified, no correlation with overall rotor entropy loss $\zeta_{s,R}$ can be detected in Fig. 5.22 and losses are outweighed by the increase in transferred power.

The influence of hot spot temperature $T_{\text{max}}^{(\text{hs})}$ on $\Delta C_u$ is less clear because mean inlet temperature $T_{t,\text{in}}$ is kept constant in all simulations. Therefore, effects of hot temperatures on NGV exit swirl angle are balanced by colder regions in the inlet traverse and their respective effects are smeared by the mixing plane. There is no distinct correlation of hot spot temperature $T_{\text{max}}^{(\text{hs})}$ with outlet temperature distribution ($|\rho(T_{\text{max}}^{(\text{hs})}, T_t(r))| < 0.5$).

### Adiabatic Wall Temperature and Heat Transfer on the Vanes

The mean values and deviations of surface temperature $T_w$ and Nusselt number Nu of run d) on the three vanes are shown in Fig. 5.24. All vanes show similar regions of high variance$^{18}$ in $T_w$ and Nu along the stagnation line and at about 20% span of the PS. These result from an influence of the stagnation line position on the structure $\bar{N}$ (see Fig. 5.17). The analysis of correlation coefficients (Appendix D.8.5) shows, that these variations are caused primarily by swirl strength $S$ and swirl clocking $\theta^{(\text{sw})}$. Only weak deviations caused directly by the trajectory of inlet swirl $\bar{B}$ can be identified towards the TE of the left vane’s PS.

Another region of high deviations, especially in heat transfer, can be identified at 30% of the left vane’s SS. It is caused by the vortex structure induced by the inlet swirl core at the LE ($\bar{C}$ in Fig. 5.17). The radial position $r_{sw}$ of this vortex is determined by the inlet swirl clocking position $\theta^{(\text{sw})}$ in accordance with the results of Section 5.2.4. The observation is confirmed by correlations$^{19}$ of $r_{sw}$ with the parameters $X$ of the inlet traverse. The radial position $r_{sw}$ is negatively correlated with the swirl clocking position ($\rho(\theta^{(\text{sw})}, r_{sw}) = -0.62^*$) in agreement with the findings from the generic inlet swirl field to the RRD-HPT in Fig. 5.9. Additionally,

---

$^{18}$ As shown in Section 3.4.3 the uncertainty in the prediction of film cooling effectiveness $\eta_{FC}$ is estimated to be about 10-20% locally. In Appendix D.8.3 it is shown that this corresponds to an uncertainty in $T_w/T_{t,\text{in}}$ of 5-10%. Values of $\sigma(T_w/T_{t,\text{in}})$ below this range are therefore not considered significant.

$^{19}$ Data from run b) are used for analysis. The position of the vortex is detected by vorticity on a plane along the stagnation line as shown in Fig. 5.17
in the E3E, a correlation with swirl number $S$ is identified ($\rho(S, r_{sw}) = -0.58^*$) which is not present in the data in Fig. 5.9.

**Figure 5.24:** Mean and standard deviation of NGV1 adiabatic wall temperature (top) and Nusselt number (bottom) from gPC run d) on the SS ($s < 0$) and PS ($s \geq 0$) of the three stator vanes. Values are computed from the gPC expansions $\tilde{Y}$. The black line marks regions of $\sigma(T_{w,ad}/T_{t,in}) > 5\%$. The data are not corrected for deviations in mean inlet temperature $T_{t,in}$. Labels correspond to the structures visualised in Fig. 5.17.

**Adiabatic Wall Temperature and Heat Transfer on the Hub End Wall**

The standard deviation of adiabatic wall temperature $T_{w,ad}$ and Nusslet number $\text{Nu}$ on the NGV hub for the four investigated cases a) to d) is shown in Fig. 5.25. It can be seen that $\sigma(T_{w,ad})$ locally reaches up to 10\% of the mean inlet temperature $T_{t,in}$. As for the vanes, only regions with $\sigma(T_{w,ad}/T_{t,in}) > 5\%$ are regarded significant. Three characteristic zones $\mathcal{P}$, $\mathcal{Q}$ and $\mathcal{S}$ of high $\sigma(T_{w,ad})$ can be clearly distinguished in all cases where a variation of inlet swirl is present (a, b, and d). They are therefore assumed to be a consequence of variance in inlet swirl number $S$ or position $\theta^{(\text{sw})}$. Since a variation of $S$ between cases a) and b) does not largely change the occurrence

5.4 Uncertainty Propagation in the Engine 3E

135
of these zones, they must be a consequence of the swirl clocking position $\theta^{\text{sw}}$ which is confirmed by the strong correlation $\rho(\theta^{\text{sw}}, T_w)$.

**Figure 5.25:** Standard deviation $\sigma$ of NGV1 adiabatic wall temperature (top) and Nusselt number (bottom) at the hub end wall (computed from the gPC expansions $\tilde{Y}$) and correlations $\rho$ with the parameters determining the inlet traverse (computed from the Monte Carlo samples $\tilde{Y}$). The black line marks regions of $\sigma(T_{w,ad}/T_{t,in}) > 5\%$ in run d). The data are not corrected for deviations in mean inlet temperature $\overline{T_{t,in}}$. 

Geometry is intentionally distorted.
Zone $\mathbb{P}$ is located primarily in the front part\textsuperscript{20} of the hub and results from an interaction of inlet swirl with the RIDN coolant (see the coolant distribution in Fig. 5.19). Zones $\mathbb{Q}$ and $\mathbb{S}$ mark regions that occur due to an interaction of inlet swirl and the vanes as they occur behind the left vane and in the passage between the central and right vane only.

Zone $\mathbb{Q}$ is the consequence of the change in PS upwash by the mechanism that is described for the RRD-HPT in Section 5.2.4 and similarly observed in the LSTR in Section 5.3.2. The radial variation of secondary flows on the PS with $\theta^{[sw]}$ occurs in analogy to the previous finding of this chapter but is only slightly visible in the deviations of $\eta_{FC}$ and Nu on the left vane ($\mathbb{B}$ in Fig. 5.24). Entrained flows below the inlet swirl core are accelerated and cause an upwash at the PS which is decreased when clocking inlet swirl ot the passage. The induced change in the radial position of residual swirl and passage secondary flows alters the migration of the hot streak, entrained into swirl, and causes it to impinge on the hub end wall in region $\mathbb{Q}$ behind the vane, as illustrated in Fig. 5.26.

In order to quantify the effect, the radial position $r_{sw}$ of the footprint of inlet swirl ($\mathbb{B}$ in Fig. 5.17) at the NGV outlet as well as hub temperatures downstream of the left and central vane are plotted over swirl strength $S$ and clocking $\Delta \theta^{[sw]}$ in Fig. 5.26 (bottom). Only the swirl clocking position $\theta^{[sw]}$ is responsible for a radial movement of the residual swirl core confirming the results of the previous sections. However, both, $\theta^{[sw]}$ and $S$, increase the hub temperature, because the increased swirl number enhances the transport of the hot streak to the hub.

Zone $\mathbb{S}$ in Fig. 5.25 results from a similar interaction of inlet swirl with the hot streak at the central vane. At this clocking position, however, the interaction between swirl, hot streak and vane inverts the correlation of swirl clocking $\theta^{[sw]}$ with wall temperature $T_{w,ad}$ of zone $\mathbb{Q}$ which illustrates the importance of the relative clocking position of inlet swirl and NGVs with respect to the hot streak migration.

Mean values and standard deviation\textsuperscript{21} of Nu in run d) are shown at the bottom of Fig. 5.25. High deviations of Nu occur mainly behind the left vane and in the passage between the central and the right vane. These correspond to areas where inlet swirl interacts with secondary flows along the vane. The deviations in Nu are much more alike in these two passages than those of $T_{w}$ which indicates that

\textsuperscript{20} The discontinuity of zone $\mathbb{P}$ in the rear part of the hub, labelled $\mathbb{P}$, correlates with a change of surface radius of the contoured end wall.

\textsuperscript{21} The heat flux $\dot{q}$ in Eqn. (2.16) is computed from an additional simulation, using a fixed wall temperature boundary condition, for each of the $N = 21$ generated traverses. The wall temperature $T_{w}$ for these simulations is in between the temperature $T_{c}$ of film cooling and the maximal temperature of the inlet traverse. Therefore, no values for Nu can be computed in the front part of the stator, where $T_{w} \approx T_{c}$, and these areas are masked in Fig. 5.25.
deviations of the hot spot are less significant for the uncertainty of heat transfer predictions.

**Figure 5.26:** Clocking inlet swirl alters its radial migration through the NGV1 passage causing a hub impingement of the hot streak behind the vane (top). Correlations of inlet swirl clocking $\theta^{\text{sw}}$ and strength $S$ with the radial position of residual swirl $r^{\text{sw}}$ at passage outlet and wall temperature $T_w$ in region $Q$ are shown at the bottom.

This observation is confirmed by the correlation coefficients at the bottom of Fig. 5.25. In the regions of high deviations behind the vanes, Nu is not significantly affected by deviations in hot spot position and temperature. As for the wall temperatures, the main factor, influencing $\sigma(Nu)$, is the swirl clocking position $\theta^{\text{sw}}$. The positive correlation $\rho(\theta^{\text{sw}}, Nu)$ behind the left vane (region $Q$) is in line with the previously discussed effects on wall temperature (Fig. 5.26). The downwash of flow on the PS, induced by clocking the inlet swirl, causes an impingement of flow behind the TE of the left vane, locally thins the boundary layer and increases heat transfer. Correlations behind the left and right vane are largely different which, again, emphasises the importance of the relative clocking of inlet conditions and vanes with regard to their uncertainty propagation.
6 Conclusion and Outlook

6.1 Summary of Key Results

The key results with regard to the research objectives, formulated in Section 1.2, are summarised here:

• *Decoupling combustor and turbine*: A major shortcoming of the numerical decoupling of combustor and turbine is the inherent discontinuity caused by the incomplete set of inlet BCs of the turbine. The distribution of velocity and static quantities at NGV1 inlet is affected and must be paid attention to as it determines the mass flow weighted average of inlet temperature. The issue is especially relevant when clocking the inlet BCs (→ Section 3.3.2).

• *Influence of turbulence diffusion on wall temperature*: The uncertainty in thermal predictions at the LSTR are governed by the modelling of turbulent diffusion to the wall which showed to have a much larger influence than different turbulence models (→ Section 3.4).

• *Modelling combustor exit flow*: Realistic 2D lean burn combustor outlet conditions can be generated from a flow model as described in this work. Realistic swirl and temperature interfaces can be matched by tuning the model parameters. The number of first order parameters, relevant in design, is small enough to be used with statistical sampling methods (→ Chapter 4).

• *Uncertainty quantification workflow*: In order to apply the model in an automated UQ framework, the deviations of inlet velocity distribution need to be corrected for. A one-step correction is the best compromise between reducing errors in the 2D distribution and imposing an erroneous drift of the averaged velocity. An inherent error introduced by the reference field cannot be fully corrected for but shows to be minor compared to characteristic variance of the inlet traverse (→ Section 4.6, Appendix C.5.8 and Section 5.3.2).

• *Statistical methods*: Uncertainty quantification by means of a polynomial chaos approach is applicable for typical problem dimensionality which increases the quantitative accuracy of the results with regard to crude Monte Carlo approaches. Nevertheless, qualitative results obtained using the two methods are equal (→ Section 5.4.5).

• *Effect of swirl on vane aerodynamics*: Regarding deviations of inlet swirl at isothermal conditions, swirl number $S$ and clocking position $\theta^{sw}$ are the most
influential parameters on stage efficiency $\eta$ and NGV pressure loss $\zeta_{NGV}$ (note, that turbulence was not varied). The dependency of $\eta$ on $S$ is non-linear and clocking positive swirl (with the same sense of rotation as the rotor) towards the PS is advantageous for pressure loss and, thus, efficiency. If swirl is clocked to the LE, at sufficiently high inlet swirl numbers the formation of secondary flow-like structures is triggered at the stagnation line, regardless of swirl orientation. The structure is identified in all three investigated geometries. Swirl clocking affects the radial position of formation of these structures as well as the radial migration of residual inlet swirl and secondary flows by an upwash from hub flow entrained into swirl. The effect outweighs swirl-induced, radial pressure gradients in the passage (→ Section 5.2).

• **Effect of swirl and hot spots on vane heat load**: The described deviations in the radial position of residual swirl and swirl-induced secondary flows are responsible for significant deteriorations of NGV hub and vane heat transfer and film cooling effectiveness at stator vanes exposed to inlet swirl. Uncertainties in NGV thermal data are therefore mostly influenced by inlet swirl clocking. The deviations on the vane are diminished if inlet swirl is clocked to the passage. The interaction of swirl and hot spots is relevant for predicted uncertainties and a combined investigation of the two is essential. Changing the relative clocking of swirl and hot spots with respect to the vanes may alter their interaction mechanisms (→ Section 5.4).

• **Effect of swirl and hot spots on stage performance**: Varying the hot spot and inlet swirl at the same time shows that swirl mostly affects NGV losses whereas the hot spot determines losses in the rotor. Stage efficiency is determined by the hot spot radial position which affects the radial distribution of rotor incidence (by preferential heating), loading and losses. The uncertainty in predicted efficiency is affected mostly by an influence on predicted power (from the local incidence), which overshadows additional production of losses in the rotor. A standard deviation of 0.17% in isentropic efficiency is found for realistic uncertainties of the 2D inlet traverse.

### 6.2 Impact of Results and Recommendations

#### 6.2.1 Global Impact on Overall Engine Efficiency

A characteristic standard deviation of HPT efficiency $\eta_{HPT}$ in the first stage\(^1\) resulting from uncertainties in the distribution of the 2D inlet conditions is quantified at 0.17% (Tab. 5.4). Turbine isentropic efficiency $\eta_{HPT}$ affects the engine’s overall

\(^1\) Note, that $\sigma(\eta_{HPT})$ of the both HPT stages is assumed to be lower.
efficiency $\eta_o$, although $\eta_o$ is mainly determined by the cycle's pressure ratio $\pi$ and the turbine inlet temperature $T_{t,40}$. The impact of the identified variance in $\eta_{HPT}$ on the engine cycle therefore depends on the ability of the manufacturer to accurately predict the other performance parameters $\pi$, $T_{t,40}$, etc. A UQ analysis of a representative engine cycle is shown in Appendix E.1. In this example, the effect of $\sigma(\eta_{HPT}) \approx 0.2$ on the uncertainty of predicted engine efficiency and thrust is overshadowed by an uncertainty of 2% in $T_{t,40}$ (which Montomoli et al. [123] give as realistic value). Better knowledge of the 2D distribution of turbine inlet condition is therefore unlikely to increase the robustness of the engine cycle prediction and designers should rather focus on the local heat load determining the life of the turbine.

6.2.2 Local Impact on Turbine Cooling and Life

Turbine metal temperature influences the creep process rate exponentially, according to Eqn. (2.14) and can drastically reduce its life time. The characteristic uncertainty in adiabatic wall temperature, quantified in this work, is between 5% and 15% at the NGV1 hub and vane end walls. Note, that these deviations of adiabatic wall temperature are extreme because heat conduction in the metal is not considered. The deviations of real, non-adiabatic wall temperatures are assumed to be lower due to the thermal inertia of the metal. They can therefore not be used for a quantitative analysis of turbine life deterioration.

Still, the large deviations indicate that a variance of the 2D distribution of lean burn HPT inlet conditions should be taken into account for a robust design of the HPT cooling system. The most influential parameters to take into account, considering the hub heat load, are the radial hot spot position $r^{(hs)}$ and the relative clocking positions of the inlet swirl core $\theta^{(sw)}$ and hot spot $\theta^{(hs)}$. With regard to the vane heat load the swirl strength $S$ and clocking position $\theta^{(sw)}$ are most influential. The mechanisms by which hot spots and swirl interact depend on their relative clocking position relative to the vanes.

6.2.3 Implications for Design

The primary degree of freedom in the design of a combustor turbine system is the clocking of the combustor relative to the turbine. The ideal clocking position is most likely determined by the position of the hot spot which can be attenuated when clocked to the vane LE (Jenkins et al. [89]) thus reducing the heat load to the rotor. With this design philosophy the swirl clocking, relative to the stator vanes, is a dependent variable for a specific combustor design. It was shown in this work in this work, turbine efficiency $\eta_{HPT}$ is considered decoupled from inlet temperature $T_{t,40}$ by keeping turbine operating conditions constant. Actually, $T_{t,40}$ has an additional, indirect influence on $\eta_o$ via $\eta_{HPT}$ which is not considered.
that, if the swirl clocking position coincides with the LE of a vane, design turbine efficiency reduction is lowest but the thermal design of the stator also becomes least robust. In this case, a UQ workflow, as developed in this work, may help to reduce risks related to uncertainties in the inlet traverse.

6.3 Outlook

An improvement of the method can be obtained by tackling any of the topics in the list of assumptions in Section 4.11. In the following, recommendations for a continuation of this work are listed, sorted from topics focused on improving the accuracy of the method to expansion of its scope:

• Systematic characterisation of 2D combustor outlet conditions, evaluation and further development of the model for traverse parameterisation on a statistical basis with more test cases of different combustor technology, geometry, operating conditions for a wider spectrum of possible turbine inlet traverses. Specific attention should be paid to the distribution of velocity in the CTI traverse and the relation of first and second order parameters used to decrease the degrees of freedom in the analysis (Tab. C.1).

• Systematic quantification of uncertainties in the CTI interface and statistical dependence between fluctuations of the temperature and swirl distribution based on a large number of different combustors. In this work, temperature and swirl fluctuations are assumed statistically independent.

• Comparison of coupled and uncoupled combustor turbine simulations, especially with regard to discontinuities across the CTI interface, by comparing the velocity distribution in the upstream and downstream traverse.

• Development of a model for the distribution of axial velocity in order to get rid of the correction loop for temperature scaling.

• Expansion of the scope of the statistical analyses, conducted in this work, towards a detailed analysis of the flow through the rotor row by means of unsteady CFD simulations enabling a consideration of transient stator-rotor interaction and the migration of hot spots in the rotor.

• Integration of deviations of turbulent quantities into the analysis.

• Consideration of unsteadiness of the flow at the CTI interface by imposing HPT inlet conditions in a time-resolved manner.

• Integration of CHT techniques into the CFD analysis in order to obtain realistic distributions of wall temperatures by taking into account the heat conduction in the metal.
A Appendix to State of the Art

A.1 Turbine Efficiency

A.1.1 Influence of Losses and Power

As was mentioned in Section 2.2.2, changes in turbine efficiency $\eta$ are not necessarily caused exclusively by changes in entropy production $\Delta s$. As shown in Fig. A.1, the efficiency of a reference process (black) can be increased by reducing $\Delta s$ (blue) but also by increasing both power $P$ and isentropic power $P_i$ (red) by $\Delta P$. The latter can be seen in the inequality

$$\frac{P}{P_i} < \frac{P + \Delta P}{P_i + \Delta P}$$

with $P_i > P > 0$ and $\Delta P > 0$. Although, strictly speaking, there is a difference between $\Delta P$ in the numerator and the denominator, due to the inclination of isobars, this relation illustrates the mechanism increasing efficiency at varying power. In the conducted studies, with varying inlet conditions, a variation of power $P$ is inevitable because deviations of both, inlet total temperature and pressure, cause deviations in circumferential flow velocity at NGV exit $C_u$ (cf. Section 2.2.3 and Section 2.4.1). According to Euler’s turbine equation (2.7) these are directly associated with a change in transferred power $P$.

A.1.2 Efficiency Computation in the Flow Solver

To the author’s experience, the process of extracting the flow quantities required for efficiency computation in CFD is not unambiguous. In order to enable comparison with other studies, efficiency definition and implementation in the used flow solver is outlined in the following. Isentropic efficiency is defined as

$$\eta = \frac{P}{P_i}$$

Figure A.1: Efficiency changes
with the power output of the real process $P$ and the isentropic process $P_i$. Following the efficiency definition by Hartsel [67], the isentropic power $P_i$ is given by the sum of the expansions of the main flow and all coolant streams

$$P_i = \sum_j \left\{ m_{in,j} c_p,j T_{t,in,j} \left[ 1 - \left( \frac{p_{t,out}}{p_{t,in,j}} \right)^{\frac{\gamma_j-1}{\gamma_j}} \right] \right\}. \quad (A.3)$$

That is, index $j$ covers the main flow inlet as well as all cooling holes (if the numerical domain is cut off within these holes) and the modelled inlets of seal and leakage flow. The mass flow $m$, total pressure $p_t$ and temperature $T_t$ on a surface $S$, required to evaluate this expression, are extracted from CFX using the built-in routines

- `massFlow()@S`
- `massFlowAve(Total Pressure in Stn Frame)@S`
- `massFlowAve(Total Temperature in Stn Frame)@S`

respectively. Averaged fluid properties $\bar{\gamma}$ and $\bar{c_p}$ are, for each individual stream $j$, extracted on the respective inlet and the domain outlet by the

- `massFlowAve()`

routine and then arithmetically averaged. The power $P$ of the real process, according to the first law of thermodynamics

$$\Delta H_t = P + \dot{Q}, \quad (A.4)$$
is equal to the enthalpy difference of the fluid $\Delta H_t$ if the process is adiabatic ($\dot{Q} \equiv 0$). On the other hand, the shaft power can be computed from the torque $M$ on all rotor blades and their rotational speed $\omega$ specified in the simulation

$$P = \omega M \quad (A.5)$$

Thus, $P$ can be computed in the CFD calculations both from Eqn. (A.4) and Eqn. (A.5). Mass flow averaged total enthalpy $H_t$ on a surface $S$ can be extracted using the built-in routine

- `massFlowInt(Total Enthalpy in Stn Frame)@S`

The torque $M_x$ on rotating surface $S$ along the $x$ axis is retrieved from

- `torque_x()@S`

In order to include the momentum of the coolant ejected from the rotating rotor blade in the momentum balance, the latter function must equally be applied to rotating solid surfaces as well as to all rotating inlets (such as those of film cooling...
holes).
In theory, both ways should yield the same value for $\eta$. In practise, however, efficiencies computed using $P = \Delta H_t$ differ from those computed using $PL = \omega M$ due to discretisation and solution errors in CFD. Efficiencies reported in this work are computed from $P = \omega M$ because it showed to be more robust against numerical fluctuations in the solution process. All efficiencies (and other integral quantities) retrieved from steady simulations are defined as an expression that is monitored for each solver iteration and averaged over multiple intervals of the respective small-scale fluctuations after the expression has reached a stable, but possibly oscillating, state.
B Appendix to Methods Review
Computational Fluid Dynamics and Uncertainty Quantification

B.1 Solving the Discretised Equations

B.1.1 Properties of the Flow Solver

The commercial flow solver CFX® [29], applied in this work, is a pressure-based solver specialised for compressible turbomachinery flows. The solution of the is coupled, i.e., the full system of equations is solved simultaneously and all variables are computed on a co-located grid in combination with an interpolation scheme similar to the proposition by Rhie and Chow [144] (as an alternative, other solvers apply a staggered arrangement of variables in order to prevent an oscillation of the pressure field). The data are stored node-centred, i.e., conservative control volumes are constructed around each mesh node by connecting the cell centres of adjacent nodes.

B.1.2 Solution of the Coupled System of Equations

Although the Navier-Stokes Equations are non-linear, solving the discretised transport equations involves a linearisation about a guessed solution (Ferziger and Peric [50]). A CFD solver therefore solves a linear system of equations

\[ Ax = b \]  

(B.1)

with a matrix of coefficients \( A \), determined by the transport equations, discretisation and mesh topology, a solution vector \( x \) and a source term \( b \) representing the domain boundaries. A direct solution by inverting \( A \) is theoretically possible but impractical for most applications due to the high numerical effort. The solution is therefore approximated by an iterative procedure, i.e., a guessed, initial solution to system (B.1) is iteratively improved. There is a number of different iterative solution methods of which a coupled incomplete LU solver is applied in the flow solver CFX. \( A \) is factored into the product of a lower and upper triangular matrix, \( L \) and \( U \), similar to a classical Gauss elimination. However, all zero-elements in \( A \) are set to zero in \( L \) and \( U \) as well, in order to obtain two system matrices equally sparse as \( A \) (therefore “incomplete” LU). System (B.1) is thus split into two coupled systems which can be efficiently solved by forward and backward elimination, respectively.
B.1.3 Multigrid Algorithm

In CFX, the solution of system (B.1) is sped up by the application of an algebraic multigrid algorithm. Most numerical solution algorithms are efficient in eliminating errors with wavelengths at the size of the mesh spacing due to their local coupling in the matrix $A$ (Schäfer [155]). In a multigrid scheme, errors of high frequency (relative to the current mesh spacing) are reduced before successively coarsening the grid after some iterations. The error components with lower frequencies are subsequently reduced on coarser grid levels before refining the grid back to the level of highest resolution. The coarsening is achieved by partially summing up the system of discrete equations on the fine mesh such that the contributions from adjacent cells merge. According to Schäfer [155], computing time is approximately proportional to mesh size using a multigrid algorithm whereas it increases progressively with mesh size otherwise.

B.2 Modelling Transient Blade Row Interaction

B.2.1 Mixing Planes

The flow through turbomachinery is often analysed assuming steady flow conditions for reasons of computational efficiency. This steady-state assumption is associated with the application of mixing-planes between the stator and rotor domains as shown in Fig. 3.2. This method enforces an infinitely fast mixing at the rotor-stator interface. Hence, circumferentially averaged flow quantities from the upstream domain serve as inlet conditions to the downstream domain. The assumption of steady-state typically results in increased solution errors from the suppression of unsteady modes. If the associated oscillations of results about a steady state are small, they can be averaged out in the post-processing (Ferziger and Peric [50]). The application of mixing planes causes modelling errors because circumferential non-uniformities, such as wakes from the trailing edges or the 2D turbine inlet conditions investigated in this work, are not resolved in the rotor domain. Apart from that, the conservativeness of the solver is challenged by the mixing process.

B.2.2 Transient Blade Row Models

In principle, TBR can be directly resolved by flow solvers. However, in order to prevent acoustic coupling, blade pitch angles between rotors and stators are usually designed such that a large number of blades must be modelled in order to be able to pass the transient information directly from the upstream to the downstream row through a sliding plane. The associated numerical effort can be reduced by scaling the blade rows to smaller or larger pitch angles or by employing TBR models that aim at resolving transient interaction at an unequal modelled sector pitch. A
number of different methods has been proposed in the literature. An overview on
the models implemented in the used commercial flow solver CFX® [29] is given by
Connell et al. [35, 36]. In this work, the time inclination method by Giles [53] is
used for transient simulations. The main idea of which is to solve the flow in the
downstream row in an inclined computational time
\[ t' = t - \frac{y \Delta T}{\theta_{\text{pitch},r}} \]
that varies in circumferential direction \( y \) depending on rotor wheel speed \( U \) and
pitch difference \( \theta_{\text{pitch},s} - \theta_{\text{pitch},r} \) between stator and rotor. Using this transformation,
each axial plane is at \( t' = \text{const.} \) which enables the usage of ordinary periodic
boundary conditions in the solver.

B.3 Numerical Simulation of Combustor Turbine Interaction

B.3.1 Conserving Enthalpy Inflow instead of Turbine Inlet Temperature

With regard to CTI, an alternative perspective on scaling the thermal conditions
at plane 40 than that described in Section 3.3.3 may be taken. This point of view
is based on an integral, thermodynamic control volume of the combined system
“combustor and turbine” as shown in Fig. B.1.

![Figure B.1: Different approaches to scaling the CTI interface conditions: A component perspective favours scaling to \( T_{t,40} \) for comparability whereas from a systemic perspective scaling to \( \dot{H}_{t,40} \) is preferable.](image)

From this point of view, no scaling of turbine inlet temperature to a common design
point is required as long as the inlet conditions to the combined system (the enthalpy
streams of fuel and compressor air to the combustor) are conserved. Variations in
\( T_{t,40} \), and thus turbine efficiency, resulting from changes in the combustor design
upstream (geometry of swirler, liner, cooling scheme, etc.) are regarded to charac-
terise an internal interaction of the two sub-systems and therefore do not need to be
corrected for. Consequently, the purpose of scaling, when simulating the turbine in isolation, is merely to conserve\(^1\) the enthalpy stream from the combustor that would reach the turbine in a coupled simulation. Hence, turbine inlet temperature is to be scaled such that
\[
\dot{H}_{t,40} = \dot{m}_i Q_R + \dot{m}_3 h_{t,3} - \dot{m}_c h_{t,c}. \tag{B.3}
\]
The distinction between conserving \(\dot{H}_{t,40}\) and \(T_{t,40}\) is only relevant if the distribution of \(T_t\) is non-homogeneous (due to mass flow averaging) and if specific heat \(c_p\) is modelled as a function of temperature. Then, fixing \(\dot{H}_{t,40}\) does not fix \(T_{t,40}\). This can be seen by comparing the definition of \(\dot{H}_{t,40}\)
\[
\dot{H}_{t,40} = \int_{40} c_p(T) T dt \neq c_p \int_{40} T dt \tag{B.4}
\]
with that of \(T_{t,40}\) in Eqn. (3.17). As shown in Appendix B.3.2, the difference in \(\dot{H}_{t,40}\) for a fixed \(T_{t,40}\) can amount to 3% between different distributions of inlet temperature. Both ways of scaling have a physical justification. In order to grant comparability with other studies, conservation of \(T_{t,40}\) is applied in this work.

### B.3.2 Variance of Enthalpy Inflow
The variability of enthalpy inflow \(\dot{H}_{t,40}\) is analysed in a thought experiment. A HPT inlet plane is idealised as a rectangle of length \(L\) and height \(H\) with area
\[
A = LH. \tag{B.5}
\]
In the following, two different inflow conditions \(\text{I}\) and \(\text{II}\) to the turbine are analysed, both with the same mass flow
\[
\dot{m} = \dot{m}_I = \dot{m}_II, \tag{B.6}
\]
axial inflow and a homogeneous distribution of mass flux and static pressure
\[
\rho u_{ax} = \text{const. and } p = p_0 = \text{const.} \tag{B.7}
\]
The gas is assumed ideal with \(R = \text{const. and } \gamma = \text{const. but with a temperature dependent heat capacity, approximated by a second order polynomial}
\[
c_p(T) = C_1 T^2 + C_2 T + C_3 \tag{B.8}
\]
where \(C_1, C_2\) and \(C_3\) are empirical constants.

\(^1\) In this case, the scaling only compensates for the difference in axial velocity between the upstream and downstream traverse described in Section 3.3.2.
Case I: Homogeneous Temperature Profile

In case I static temperature is assumed to be distributed homogeneously, thus

\[ T_1 = T_0 = \text{const.,} \quad \rho_1 = \frac{P_0}{RT_0}, \quad u_{ax,1} = \frac{\dot{m}}{\rho_1 A} = \frac{\dot{m} RT_0}{Ap_0} \quad (B.9) \]

and the total temperature can be computed as

\[ T_{t,1} = T_0 + \frac{u_{ax,1}^2}{c_p(T_1)} = T_0 + \frac{(\dot{m} RT_0/Ap_0)^2}{C_1 T_0^2 + C_2 T_0 + C_3}. \quad (B.10) \]

Since all quantities in this expression are homogeneous, this temperature is equal to the mass flow averaged total temperature \( \overline{T_{t,1}} \) for this case

\[ \overline{T_{t,1}} = \frac{1}{\dot{m}} \int \int T_{t,1} \rho_1 u_{ax,1} \, dx \, dy = T_0 + \frac{(\dot{m} RT_0/Ap_0)^2}{C_1 T_0^2 + C_2 T_0 + C_3}. \quad (B.11) \]

The inflow of total enthalpy \( \dot{H}_{t,1} \) is therefore given by

\[ \dot{H}_{t,1} = \int \int c_p(T_1) T_{t,1} \rho_1 u_{ax,1} \, dx \, dy = \dot{m} \left[ (C_1 T_0^2 + C_2 T_0 + C_3)T_0 + \left(\frac{\dot{m} RT_0}{Ap_0}\right)^2 \right]. \quad (B.12) \]

Case II

In case II a non-homogeneous temperature distribution is assumed with a hot zone of size \( A_h = \alpha A \) and temperature \( T_h \) and a cold zone of size \( A_c = (1 - \alpha)A \) and temperature \( T_c \) according to Fig. B.2. All quantities are assumed to be distributed homogeneously within the two zones and referenced by indices \( \square_c \) and \( \square_h \), respectively. The flow quantities in these zones can then be computed as follows

\[ \dot{m}_c = (1 - \alpha)\dot{m}, \quad \dot{m}_h = \alpha \dot{m}, \]
\[ \rho_c = \frac{P_0}{RT_c}, \quad \rho_h = \frac{P_0}{RT_h}, \]
\[ u_{ax,c} = \frac{\dot{m} RT_c}{Ap_0}, \quad u_{ax,h} = \frac{\dot{m} RT_h}{Ap_0}, \]
\[ T_{t,c} = T_c + \frac{(\dot{m} RT_c/Ap_0)^2}{C_1 T_c^2 + C_2 T_c + C_3}, \quad T_{t,h} = T_h + \frac{(\dot{m} RT_h/Ap_0)^2}{C_1 T_h^2 + C_2 T_h + C_3}. \quad (B.13) \]
The mass flow averaged total temperature \( \overline{T_{t,\text{II}}} \) of the inlet traverse for case (II) is given by

\[
\overline{T_{t,\text{II}}} = (1 - \alpha)T_{t,c} + \alpha T_{t,h} = (1 - \alpha) \left[ T_c + \frac{(\dot{m}\mathcal{R}T_c/Ap_0)^2}{C_1 T_c^2 + C_2 T_c + C_3} \right] + \alpha \left[ T_h + \frac{(\dot{m}\mathcal{R}T_h/Ap_0)^2}{C_1 T_h^2 + C_2 T_h + C_3} \right]
\]  

(B.14)

and the inflow of total enthalpy \( \dot{H}_{t,\text{II}} \) is given by

\[
\dot{H}_{t,\text{II}} = (1 - \alpha)\dot{m} \left[ (C_1 T_c^2 + C_2 T_c + C_3)T_c + \left( \frac{\dot{m}\mathcal{R}T_c}{Ap_0} \right)^2 \right] + \alpha \dot{m} \left[ (C_1 T_h^2 + C_2 T_h + C_3)T_h + \left( \frac{\dot{m}\mathcal{R}T_h}{Ap_0} \right)^2 \right].
\]  

(B.15)

**Difference in Enthalpy Inflow for Equal Mean Total Temperatures**

For the analysis of differences in total enthalpy inflow \( \Delta \dot{H}_t \) between the two cases, it is postulated that both cases have the same mass flow averaged total temperature \( \overline{T_{t,\text{I}}} = \overline{T_{t,\text{II}}} \).

Thus, different combinations of cold and hot temperature, \( T_c \) and \( T_h \), in case (II) can be derived that have the same mass flow averaged total temperature as case (I). The equation

\[
C_1 T_h^3 + \left[ C_2 - \frac{\overline{T_{t,\text{I}}}}{\alpha} - (1 - \alpha)T_{t,c} \right] C_1 + \left( \frac{\dot{m}\mathcal{R}}{Ap_0} \right)^2 T_h^2 + \left( C_3 - \frac{\overline{T_{t,\text{I}}}}{\alpha} - (1 - \alpha)T_{t,c} \right) C_2 T_h - \frac{\overline{T_{t,\text{I}}}}{\alpha} C_3 = 0
\]  

(B.17)

determines the hot zone temperature \( T_h \) in case (II) for a given cold zone temperature \( T_c \), area ratio \( \alpha \) and mean temperature \( \overline{T_{t,\text{I}}} \) of case (I). This equation is solved for several parameter combinations in order to produce different traverses of case (II) with equal mass flow averaged temperatures \( \overline{T_t} \) as case (I).

The following set of parameters is used which is arbitrary but representative for cruise conditions in a turbofan engine:

\[
\begin{align*}
H &= 0.1 \text{ m}, & T_0 &= 1600 \text{ K}, & T_c &\in [1200 \text{ K}, \ 1600 \text{ K}], \\
L &= 0.1 \text{ m}, & p_0 &= 8 \times 10^5 \text{ Pa}, & \gamma &= 1.33, \\
\alpha &\in [10\%, \ 50\%], & \dot{m}_0 &= 0.625 \text{ kg/s}, & \mathcal{R} &= 287 \text{ J/kg K}, \\
C_1 &= 3.3 \times 10^{-5} \text{ J K/kg}, & C_2 &= 0.1 \text{ J/kg}, & C_3 &= 1000 \text{ J/kg K}.
\end{align*}
\]  

(B.18)
The cold zone temperature $T_c$ as well as the area ratio $\alpha$ are varied within the given range. Results are shown in Fig. B.3. A considerable variation of $\Delta H_{t,\text{in}}$ can be observed and for a small hot zone at $\alpha = 0.1$ the relative difference in $\Delta H_{t,\text{in}}$ grows fast for decreasing cold zone temperature $T_c$. However, if $T_h$ is limited to a realistic regime below 2200 K, $\Delta H_{t,\text{in}}$ is below 1%. For larger hot zones with $\alpha > 0.4$, however, $\Delta H_{t,\text{in}}$ can reach values of up to 3% even in this realistic regime of $T_h$.

![Graph showing solutions of Eqn. (B.17)](image)

**Figure B.3:** The plot on the left shows solutions of Eqn. (B.17), i.e. hot zone temperatures $T_h$ for given cold zone temperatures $T_c$ and area ratios $\alpha$, for which the mean temperature of case II is equal to that of case I (1600 K). The right plot shows the corresponding relative differences of total enthalpy inflow between case I and II for varying $T_c$. The dotted parts of these curves indicate a regime with unrealistically high hot zone temperatures of $T_h > 2200$ K.

### B.4 Sampling Techniques

#### B.4.1 Hammersley Sampling

The content of this section is reproduced from LaValle [108]. *Hammersley Sampling* is a *low discrepancy* sampling method (the discrepancy is a measure of how much a point cloud deviates from a balanced distribution). The method aims to achieve the best possible coverage of a probability space with few sampling points by avoiding a clustering of points that can be observed with simple random sampling.

$N$ samples $\mathbf{x}$ in a $n$-dimensional probability space are constructed from natural numbers $q = 0, 1, 2, \ldots, N - 1$. A sampling point $r(i,p)$ is constructed by first re-writing the $i$th natural number in a $p$-based representation

$$q = C_0 + pC_1 + p^2C_2 + p^3C_3 + \ldots \quad \text{(B.19)}$$

In this notation, the number 517₁₀, for instance, would be represented by $517₁₀ = 7 + 10 \times 1 + 100 \times 5$ in the decimal system (with base $p = 10$) and by $517₁₀ =$
$1000000101_2 = 1 + 1 \times 2^2 + 1 \times 2^9$ in the binary system ($p = 2$). The order of the bits is then reversed and the decimal point is moved to the left by one position such that

$$r(i,p) = C_0/p + C_1/p^2 + C_2/p^3 + C_3/p^4 + \ldots.$$  \hspace{1cm} (B.20)

A Hammersley sequence is now constructed from $n$ different bases $p_1 = 2, p_2 = 3, \ldots, p_{n-1} = n$. The $i^{th}$ $n$-dimensional sample point $x_i$ is then given by

$$x_i = [i/N, r(i, p_1), \ldots, r(i, p_{n-1})].$$  \hspace{1cm} (B.21)

![Figure B.4: Voronoi diagramm illustrating the distribution of $N = 196$ sampling points in $n = 2$ dimensions. Pseudo-random sampling shows an unwanted clustering of sample points (red cells) which thus do not add new information in the probabilistic analysis (high discrepancy). With low-discrepancy Hammersley sampling this clustering does not occur (adapted from LaValle [108]).](image)

**B.4.2 Latin Hypercube Sampling**

The *Latin-Hypercube Sampling* (LHS) method improves the reproduction of the PDFs at a low number of sampling points $N$ (Voigt [177]). The principle of LHS is illustrated in Fig. B.5 for two parameters $X_1$ and $X_2$. The stochastic domain is divided into subdomains of varying sizes according to the CDFs $F(X)$. The CDFs $F(X)$ are therefore divided into slices of equal width which are projected onto the corresponding $X$. Then, within exactly one subdomain per row and column a sample point is placed using a uniform random number generator. Thus, many sampling
points are placed in regions of high probability density of the parameters $X$ and fewer points in regions of low density.

![Diagram showing 2D Latin-Hypercube Sampling](image)

**Figure B.5**: 2D Latin-Hypercube Sampling (adapted from Voigt [177])

### B.5 Polynomial Chaos Expansions - Quadrature Methods

Quadrature methods make use of the orthogonality condition on the used polynomials $\Psi_k(\xi)$ that is projected onto the basis $\langle \Psi_i, \Psi_j \rangle = \delta_{ij} \langle \Psi_i^2 \rangle$ which yields the expression

$$\gamma_k = \frac{\langle y \Psi_k \rangle}{\Psi_k^2} \tag{B.22}$$

for the required coefficients (Najm [127]). The unknown to be evaluated to retrieve $y_k$ is the numerator

$$\langle y \Psi_k \rangle = \int y(\xi) \Psi_k(\xi) d\xi. \tag{B.23}$$

This integral can be evaluated numerically by quadrature methods. The sample points for the integration are provided by $N$ evaluations of the probabilistic system (3.23). The required number of model evaluations, i.e., CFD runs, $N$ scales exponentially with the number of uncertain parameters $n$

$$N = (p + 1)^n \tag{B.24}$$

which is why quadrature methods are suitable only for problems with very few parameters $n$. 
B.6 Verification of Uncertainty Quantification Methods

B.6.1 Cycle Properties of Example Problem

The engine data used to illustrate the Monte Carlo method in Section 3.5.4 are taken from Problem 4.13 in Farokhi [48], *Cycle analysis of a turbofan engine*. The data used are given in Tab. B.1.

| Table B.1: Characteristic Cycle Properties of Turbofan Engine |
|-----------------|------------------|------------------|
| **Ambient Conditions** |                 |                  |
| Flight Mach number $Ma_\infty$ | 0.88             | Ambient Pressure $p_\infty$ | 15 kPa |
| Ambient Temperature $T_\infty$ | $-40^\circ$C     |                  |
| **Fan and Compressor** |                 |                  |
| Design Bypass Ratio $\alpha$ | 8.0              | HP Compressor Pressure Ratio $\pi_{\text{HPC}}$ | 24.874 |
| Intake Pressure Ratio $\pi_d$ | 0.995            | Polytropic Inner LPC Efficiency $\eta'_{\text{LPC},i}$ | 70.0% |
| Inner Fan Pressure Ratio $\pi_{f,i}$ | 1.6            | Polytropic Outer LPC Efficiency $\eta'_{\text{LPC},o}$ | 90.0% |
| Outer Fan Pressure Ratio $\pi_{f,o}$ | 1.6            | Polytropic HPC Efficiency $\eta_{\text{HPC}}$ | 90.0% |
| Bypass Duct Pressure Ratio $\pi_{\text{fn}}$ | 0.95           |                  |
| **Burner** |                 |                  |
| Burner Exit Temperature $T_{t,40}$ | 1624.5 K         | Burner Design Efficiency $\eta_b$ | 99.2% |
| Fuel Heating Value $Q_R$ | 42 MJ/kg         | Burner Pressure Ratio $\pi_b$ | 0.95 |
| **Turbine** |                 |                  |
| Isentropic HPT Efficiency $\eta_{\text{HPT}}$ | 90.0% | HP Spool Mechanical Efficiency $\eta_{\text{m,HP}}$ | 95.0% |
| Polytropic LPT Efficiency $\eta_{\text{LPT}}$ | 84.0% | LP Spool Mechanical Efficiency $\eta_{\text{m,LP}}$ | 95.0% |
| Turbine Exit Duct Pressure Ratio $\pi_n$ | 98.0% |                  |

B.7 Post-processing of Polynomial Chaos Analyses

In the analysis of turbine CFD results one is typically interested in distributions of 2D data such as temperatures and heat transfer coefficients on the metal walls. The gPC analysis described in Section 3.5.5 can be readily applied to 2D data by considering the condition of each surface mesh point of the numerical grid as an uncertain parameter $Y$. That is, the expansion (3.28) is set up and the system (3.31) is solved for each mesh point.

As described in Section 3.5.5, there are two parameters determining the numerical procedure of gPC: The order of the used polynomials $p$ and the number of collocation points $N$ (corresponding to a certain oversampling ratio $n_p$) which determines how many simulations are run in order to generate samples for the system of equations (3.31). Also, the stability of the regression scheme for the solution of the system (3.31) affects the results.
The LSTR NGV1, analysed in Section 3.4, serves as an example to illustrate the effects of these parameters on the method. The swirl clocking position and strength of the swirl inlet condition are varied as uncertain input, i.e.,

\[ \mathbf{X} = [\theta^{(sw)}, S] \]  \hfill (B.25)

and the standard deviation of film cooling effectiveness (2.17) is evaluated as uncertain output

\[ \mathbf{Y} = \eta_{FC}. \]  \hfill (B.26)

Details of the process can be found in Section 5.3.2. Results obtained using different combinations of parameters are shown in Fig. B.6. The contours on the left show the uncertainty in film cooling effectiveness \( \sigma(\eta_{FC}) \) from three different gPC runs at increasing polynomial order \( p \) between 1 and 3 and a fixed oversampling ratio of \( n_p = 2 \). According to Eqn. (3.32) this corresponds with an increase of required model evaluations \( N \) between 6 and 20.

Contours on the right show results of the gPC run with \( N = 20 \) using polynomials of increasing order \( p \) between 1 and 4. In this case, the oversampling ratio \( n_p \) varies between 1.33 and 6.67 because \( N \) is fixed.

<table>
<thead>
<tr>
<th>gPC</th>
<th>Increasing Number of Samples</th>
<th>Constant Number of Samples</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>( p = 1 ) ( p = 2 ) ( p = 3 )</td>
<td>( p = 1 ) ( p = 2 ) ( p = 3 ) ( p = 4 )</td>
</tr>
<tr>
<td></td>
<td>( n_p = 2 ) ( n_p = 6.67 ) ( n_p = 3.33 ) ( n_p = 2.00 ) ( n_p = 1.33 )</td>
<td></td>
</tr>
<tr>
<td>Least squares</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ridge regression</td>
<td></td>
<td></td>
</tr>
<tr>
<td>MCS</td>
<td>( N = 6 ) ( N = 12 ) ( N = 20 )</td>
<td>( N = 20 )</td>
</tr>
</tbody>
</table>

\[ 0 \leq \sigma(\eta_{FC}) \leq 10 \]

**Figure B.6:** Standard deviation of film cooling effectiveness computed using gPC (top) and MCS (bottom). Polynomial expansions of different orders \( p \) are compared. The number of simulations \( N \) used to compute the gPC results on the left is determined from Eqn. (3.32) at \( n_p = 2 \). The gPC results on the right are computed from \( N = 20 \) (allowing for a comparison of \( p \)). Also, two regression methods for solving the system of equations (3.31) are compared.

In all gPC runs, system (3.31) is solved using both, ordinary least squares and a ridge regression scheme. The crude Monte Carlo results shown at the bottom serve...
as reference. The following conclusions can be drawn from the analysis of the gPC process:

- A ridge regression scheme provides a more robust solution of the system (3.31) than ordinary least squares regression. Therefore, ridge regression is applied in the gPC post-processing of this work.

- The solution of system (3.31) becomes unstable for high polynomial degrees $p$. Using ridge regression, results remain stable up to a degree of $p = 3$ whereas deviations from the MCS reference with least squares regression are visible already at $p = 2$. The best agreement between gPC and MCS results is in fact obtained for $p = 1$ for this low-dimensional case. Uncertainty of $\eta_{FC}$ is predicted larger with $p > 1$ than in the MCS reference case. A polynomial degree of $p = 2$ is used in this work because the possible instabilities outweigh the increase in accuracy at higher degrees.
Appendix to Method Development
Combustor Exit Flow Model

C.1 Modelling of the Velocity Field

C.1.1 Lamb-Oseen Vortex

The Lamb-Oseen vortex used to model combustor exit swirl describes an unbounded, viscous vortex of strength $\Gamma$ by blending a solid body rotation in the core with diameter $D^{\text{sw}}$ into a potential vortex in the outer region. The properties of solid body rotation, a potential vortex, the Lamb-Oseen model and the influence of both parameters, $\Gamma$ and $D^{\text{sw}}$, on swirl velocity $u^{\text{sw}}$ are illustrated in Fig. C.1.

![Figure C.1: Lamb-Oseen vortex](image)

C.1.2 Swirl Number Definition

The swirl number $S_0$, Eqn. (2.2), is defined only for axisymmetric problems such as the one shown in the top row of Fig. C.2. However, in the context of this work it is desirable to define a similar quantity $S$ for turbine inlet sectors with a geometry as shown in the centre row of Fig. C.2. Since this problem domain is not axisymmetric, a swirl number $S_0$ cannot be computed. Instead, a pseudo-swirl number $S$ is computed, in the context of this work, by integrating the fluxes $G$ over the entire interface area using the local radius $R$ as shown in centre row of Fig. C.2. The quantity $S$ is not comparable with $S_0$ because swirl momentum is weighted more for large $R$, i.e., $S$ deviates from $S_0$ for geometries with wide sectors. Therefore, the pseudo-swirl number $S$ of different configurations is only comparable if the sector geometry is unchanged.

As described in Section 4.3.1, the swirl core is circumferentially stretched in order to better match realistic flow conditions behind annular combustors. The definition of
S is hence adjusted and the radial coordinate $R^*$ is used which is measured from the closest swirl centre of the stretched vortex to a given point (cf. Fig. 4.3). The swirl centre is thus stretched to a line (approximately). Note, that $S$ is not independent of swirl stretching as can be seen in Fig. C.2 and $S$ and the swirl stretching parameter $\chi^{[sw]}$ cannot be varied at the same time. If the periodic swirl field is clocked, the radius $R^*$ is clocked likewise and defined by the ghost vortices of the neighbouring sectors (Fig. 4.4) in the clocked regions of the sector as shown in the bottom row of Fig. C.2.

**Swirl Number Definition**

<table>
<thead>
<tr>
<th>Flow Field</th>
<th>R-Isolines</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Diagram" /></td>
<td><img src="image2.png" alt="Diagram" /></td>
</tr>
</tbody>
</table>

**Figure C.2:** Definition of the pseudo swirl number $S$

**C.1.3 Modelling of Cross Flow from Converging End Walls**

The cross flow from the hub which results from the inclination of the hub end wall is modelled by the function

$$\tilde{f}(h_{rel}) = f(h_{rel})/\max[f(h_{rel})]$$

with $f_h(h_{rel}) = \left\{ 1 - \exp \left[ -\left( \frac{1-h_{rel}}{\zeta_{h}^{[ec]}} \right)^{\psi_h^{[ec]}} \right] \right\} \exp(h_{rel})$  \quad (C.1)
with the two parameters $\zeta^{(ec)}_h$ and $\psi^{(ec)}_h$. An analogue, reversed function $f_s(h_{rel})$ with $\zeta^{(ec)}_s$ and $\psi^{(ec)}_s$ is used for modelling the cross flow from the shroud.

### C.1.4 Modelling of Flow Entrainment

As described in Section 4.3.3, additional flow components are added to the modelled swirl velocity field in order to account for the effect of entrainment. These additional flow components are visualised by subtracting the swirl field from the overall traverse and dividing it into a radial and a circumferential component as shown in Fig. C.6 together with schematic sketches of these flow components. The flow components can be further divided into a contribution from the hub and shroud, respectively, as described in Section 4.3.3. The modelling of the hub contribution to both components is described in the following, the contribution from the shroud is modelled similarly.

**Distribution in $\theta$-Direction**

The distribution in $\theta$-direction of the radial and circumferential component of flow entrainment is qualitatively equal: A maximum can be identified at a certain $\theta$-location that flattens out approximately symmetrically in positive and negative $\theta$-direction. The $\theta$-distribution of the radial component $g_{rad,hl}(\theta)$ and the tangential component $g_{tan,hl}(\theta)$ are therefore modelled by the same function $g(\theta)$ with different input parameters

\[
g_{rad,hl}(\theta) = \left[ \sin\left(\frac{\theta^* + \pi}{2} + \left(\Delta \theta^{(fe)}_{rad,hl} - (\theta_{sb,hl} - \theta_{min}) \frac{2\pi}{\theta_{pitch}}\right)\right) + 1 \right] \psi^{(fe)}_{rad,hl}
\]

\[
g_{tan,hl}(\theta) = \left[ \sin\left(\frac{\theta^* + \pi}{2} + \left(\Delta \theta^{(fe)}_{tan,hl} - (\theta_{sb,hl} - \theta_{min}) \frac{2\pi}{\theta_{pitch}}\right)\right) + 1 \right] \psi^{(fe)}_{tan,hl}
\]

where $\theta_{min} < \theta < \theta_{max}$ and $\theta^* = 2\pi \frac{\theta - \theta_{min}}{\theta_{max} - \theta_{min}}$.

The function $g_{rad|tan,hl}(\theta)$ is one period of a sine wave that is scaled to have a single maximum $\max[g_{rad|tan,hl}(\theta)] = 1$ and a minimum $\min[g_{rad|tan,hl}(\theta)] = 0$. 

![Figure C.3: Function $g$](image)
The circumferential position of the maximum is determined by the four parameters \( \Delta \theta \) \( \{ \text{fe} \} \). These are defined relative to the boundaries of the swirl core

\[
\theta_{sb,h} = \theta_{mid} + \Delta \theta \{ \text{sw} \} \pm d \{ \text{sw} \} \left( \frac{D \{ \text{sw} \}}{2} \right)^{d \{ \text{sw} \}} \frac{2 \pi r \{ \text{sw} \}}{N_{sec}} \theta_{pitch}.
\]

(C.3)

Therefore, if the swirl core is moved by a change in the parameter \( \Delta \theta \) \( \{ \text{sw} \} \), the location of entrainment “follows”.

The exponents \( \psi \) \( \{ \text{fe} \} \) \( \{ \text{rad,tan} \} \) determine, how quickly the peak flattens out in \( \theta \)-direction.

The parameters may be set differently for the radial and the circumferential component and, for each component, apply both for the hub and shroud part. Thus, in total there are eight parameters used for modelling the four functions \( g_{\text{rad,h}}(\theta) \), \( g_{\text{rad,s}}(\theta) \), \( g_{\text{tan,h}}(\theta) \) and \( g_{\text{tan,s}}(\theta) \).

Distribution in Radial Direction

Unlike the \( \theta \)-distribution, the \( r \)-distribution of the circumferential and radial entrained flow components is modelled by two different functions, \( f_{\text{hl}}(h_{\text{rel}}) \) and \( h_{\text{hl}}(h_{\text{rel}}) \), respectively.

Radial Component

The peak of the radial component lies within the flow sector (not directly at the hub as for the circumferential component). Its \( r \)-distribution must therefore be modelled by a function that is zero at hub and shroud and has a single maximum in between. Also, the radial position of the peak must be controllable by a parameter \( \zeta \) \( \{ \text{fe} \} \) \( \{ \text{rad} \} \). These requirements are satisfied by the function \( \tilde{h}(h_{\text{rel}}) \)

\[
\tilde{h}(h_{\text{rel}}) = \frac{h_{\text{hl}}(h_{\text{rel}})}{\max[h_{\text{hl}}(h_{\text{rel}})]},
\]

where \( h_{\text{hl}}(h_{\text{rel}}) = (1 - h_{\text{rel}}) \left[ 1 - \exp \left( -\frac{h_{\text{rel}}}{C} \right) \right] \).

(C.4)

The parameter \( \zeta \) \( \{ \text{fe} \} \) \( \{ \text{rad} \} \) controlling the radial position of the peak is mapped to a constant \( C \) in order to enable a more tangible domain of definition of \( \zeta \) \( \{ \text{fe} \} \) \( \{ \text{rad} \} \) \( \in [0, 1] \). The two parameters are related by

\[
\zeta = 2C \left[ f_{\text{W}} \left( \exp \left( 1 + \frac{1}{C} \right) \right) \right] - 2C - 1,
\]

(C.5)
where $f_W(x)$ is the Lambert $W$ function.

**Circumferential Component**

The circumferential component has its peak close to the hub end wall and decays into the domain. Its radial shape is therefore modelled by a simple exponential function $\tilde{f}_h(h_{rel})$

$$\tilde{f}_h(h_{rel}) = \frac{f_h(h_{rel})}{\max[f_h(h_{rel})]},$$

where $f_h(h_{rel}) = \exp(-2h_{rel})$.  

Figure C.5: Function $f$
Figure C.6: Functions $f$, $g$ and $h$ used to model components of entrained flow at the hub.
C.2 Modelling of the Pressure Field

C.2.1 Effects of Static Pressure and Mach Number on Total Pressure Variation

The total pressure $p_t$ in the traverse can be expressed in terms of its static pressure $p$ and Mach number $Ma$

$$p_t = p + \frac{\rho}{2} u^2 = p \left(1 + \frac{1}{2RT} u^2\right) = p \left(1 + \frac{\gamma}{2} Ma^2\right). \quad (C.7)$$

This expression is unusual, as $Ma$ is used together with the assumption of incompressibility, but it is used here due to the non-dimensional formulation of velocity. A variation in normalised total pressure $\Delta p_t/p_t$ due to a variation in static pressure $\Delta p/p$ is of order $O(0.01)$ at a Mach number $Ma = O(0.1)$ can be estimated to be of order $O(0.01)$ as well

$$\frac{\Delta p_t}{p_t} \approx \frac{\Delta p}{p} = O(0.01). \quad (C.8)$$

A corresponding total pressure variation due to a variation in Mach number $\Delta Ma \approx Ma = O(0.1)$ can be approximated as follows

$$p_{t, Ma+\Delta Ma} = p \left[1 + \frac{\gamma}{2} (Ma + \Delta Ma)^2\right] = p_t + \gamma p Ma \Delta Ma + p \frac{\gamma}{2} (\Delta Ma)^2 \quad (C.9)$$

$$\Rightarrow \frac{\Delta p_t}{p_t} \left|_{\Delta Ma} \right. = \frac{\gamma p Ma \Delta Ma + p \frac{\gamma}{2} (\Delta Ma)^2}{p_t} \approx \frac{3}{2} \gamma (\Delta Ma)^2 = O(0.01). \quad (C.10)$$

Thus, variations in total pressure resulting from characteristic variations in static pressure $\Delta p/p = O(0.01)$ and Mach number $Ma = O(0.1)$ are of the same order.

C.2.2 Modelling of the 2D Pressure Profile

The shape of the pressure profile $p(r, \theta)$ can be derived from a known velocity distribution $u(r, \theta)$ if one assumes that gradients in momentum are balanced by a gradient in pressure. With this assumption, one can compute the pressure gradient $\partial p/\partial r$ from the Euler momentum equation

$$\frac{\partial p}{\partial r} = -\rho \left[u_{rad} \frac{\partial u_{rad}}{\partial r} + \frac{1}{r} \left(u_{tan} \frac{\partial u_{rad}}{\partial \theta} - u_{tan}^2\right)\right] \quad (C.11)$$

if unsteadiness, gradients in axial direction and viscous effects are neglected. In order to obtain the pressure field $p(r, \theta)$ from its gradient $\partial p/\partial r$, the pressure level must be specified at one radial position, for instance at the hub $p_h = p(r_h)$. The specification of the pressure level is arbitrary as long as Eqn. (4.2) is fulfilled. It is
therefore assumed that the mean total pressure at each radial band with $\theta = \theta_0$ is equal to the design total pressure
\[
\frac{\int_{r_h}^{r_s} p_t(r, \theta = \theta_0) \, dr}{r_s - r_h} = p_{t,DP}.
\] (C.12)

C.3 Blending of the Temperature Profile

As described in Section 4.7.3, combustor wall film cooling from a reference temperature field $T^{(\text{ref})}$ is blended into the modelled temperature field $T$ yielding a combined field

\[
\tilde{T}(r, \theta) =
\begin{cases}
T^{(\text{ref})}(h_{\text{rel}}, \theta) & \text{if } h_{\text{rel}} < h_{\text{rel},h}^* \text{ or } h_{\text{rel}} > h_{\text{rel},s}^* \\
\frac{h_{\text{rel}} - h_{\text{rel},h}^*}{\Delta h} T(h_{\text{rel},h}^* + \Delta h, \theta) + \frac{h_{\text{rel},h}^* - h_{\text{rel}} + \Delta h}{\Delta h} T^* & \text{if } h_{\text{rel},h}^* < h_{\text{rel}} < h_{\text{rel},h}^* + \Delta h \\
\frac{h_{\text{rel},s} - h_{\text{rel}}}{\Delta h} T(h_{\text{rel},s}^* - \Delta h, \theta) + \frac{h_{\text{rel}} - h_{\text{rel},s} + \Delta h}{\Delta h} T^* & \text{if } h_{\text{rel},s}^* - \Delta h < h_{\text{rel}} < h_{\text{rel},s}^* \\
T(h_{\text{rel}}, \theta) & \text{if } h_{\text{rel},h}^* + \Delta h \leq h_{\text{rel}} \leq h_{\text{rel},s}^* - \Delta h
\end{cases}
\] (C.13)

where $h_{\text{rel},h}^*$ and $h_{\text{rel},s}^*$ are those points, closest to the hub and shroud, respectively, where $T^{(\text{ref})} = T^*$ for a given $\theta$.

C.4 Linking of First Order and Second Order Parameters

As was shown in Section 4.9, the number of model parameters is reduced for practical applications by expressing the dependent second order parameters of the swirl field

\[
\Phi_{20} = \left[u_{\text{rad},h|s}^{(fe)}, u_{\text{rad},h|s}^{(fe)}, \Delta \theta_{\text{rad}|\text{tan},h|s}^{(fe)}, \delta_{\text{rad},h|s}^{(fe)}\right]
\] (C.14)

as functions of the first order parameters $\Phi_{10}$. The respective functions and conditions are given in Tab. C.1 for each parameter together with the respective underlying assumptions. The numerical values in the equations are determined from the 2D traverse of the LSTR by applying the matching process described in Section 4.10.1 and rounding results to characteristic values.
Table C.1: Linking of First and Second Order Parameters

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Equation/condition</th>
<th>Assumption</th>
</tr>
</thead>
<tbody>
<tr>
<td>( u_{\text{rad},h}^{{\text{fe}}} )</td>
<td>( = 3 \max[u_{\text{rad}}^{{\text{sw}}}] )</td>
<td>Magnitude of radial entrainment is assumed to depend linearly on swirl strength*</td>
</tr>
<tr>
<td>( u_{\text{rad},s}^{{\text{fe}}} )</td>
<td>( = 2 \max[u_{\text{rad}}^{{\text{sw}}}] )</td>
<td></td>
</tr>
<tr>
<td>( u_{\text{tan},h}^{{\text{fe}}} )</td>
<td>( \Leftarrow u_{\text{tan}}(\theta_{sb}, r_{h}) = 0 )</td>
<td>Tangential velocity at hub/shroud is zero at outer borders of swirl core ( \theta_{sb} ) (Fig. C.7).</td>
</tr>
<tr>
<td>( u_{\text{tan},s}^{{\text{fe}}} )</td>
<td>( \Leftarrow u_{\text{tan}}(\theta_{sb}, r_{s}) = 0 )</td>
<td></td>
</tr>
<tr>
<td>( \Delta \theta_{\text{rad</td>
<td>tan</td>
<td>h</td>
</tr>
<tr>
<td>( \zeta_{\text{rad},h}^{{\text{fe}}} )</td>
<td>( = r^{{\text{sw}}} - 0.5 )</td>
<td>Radial maximum of entrained flow moves up/down with swirl centre*</td>
</tr>
<tr>
<td>( \zeta_{\text{rad},s}^{{\text{fe}}} )</td>
<td>( = r^{{\text{sw}}} + 0.5 )</td>
<td></td>
</tr>
</tbody>
</table>

* numbers are determined from LSTR traverse

The boundaries of the swirl core \( \theta_{sb} \) are defined in Eqn. (C.3). The equations and conditions used for linking first and second order parameters are ranked by the strength of the underlying assumptions and their impact on turbine flow:

- Assuming the magnitude of radial entrainment to grow linearly with swirl strength/maximum radial swirl angle \( \max[u_{\text{rad}}^{\{\text{sw}\}}] \) is an assumption with little physical justification which is made only due to a lack of reliable data. A non-linear relation to multiple parameters of the flow and interface geometry is likely. Also, a relevant impact of this relation on turbine flow is expected. Therefore, the assumption should be reassessed based on a broader data base in the future.

- Assuming the zero tangential velocity location in Fig. C.7 to be aligned approximately with the borders of the swirl core is physically justified because at this location the radial momentum induced by swirl is highest. However, the alignment of both positions is an idealisation of the real flow field which is why the assumption is assigned a medium strength.

- The assumption of flow entrainment being clocked at a constant offset to the inlet swirl core is weak as it is fundamental to all clocking studies of turbine inlet conditions.

- Assuming the radial positions of highest magnitude of entrained flow to move up and down along with the swirl centre is an assumption of little physical justification but, at the same time, of small physical impact on the turbine as the results of this work show.
Figure C.7: Determining the magnitude of circumferential entrained flow. The magnitude is increased until the circumferential velocity of the overall flow field at the circumferential positions $\theta_{sb}$ is zero, i.e., the flow has no circumferential component at the outer borders of the swirl core.

C.5 Evaluation of Combustor Exit Flow Model

C.5.1 Matlab Genetic Algorithm

Probabilistic optimisers may be roughly categorised as gradient based or evolutionary. Since the former class is prone to finding local optima instead of the global optimum (Chipperfield et al. [33]), an algorithm of the second class is used in this work. The MATLAB® [173] genetic algorithm provides a global search method which is direct, i.e., not based on surrogate models. The optimiser mimics the natural process of selection in evolution by applying the principle of survival of the fittest. Several generations of a population of $N_{\text{pop}} = 30$ to 100 individuals are generated. Each individual is assigned a chromosome, representing a set of parameters $X$. In nature, individuals with “good genes” are more likely to reproduce by outplaying their rivals. This principle of evolution is mimicked by the optimiser by assigning individuals with high fitness, i.e., low $Y$, a higher probability for reproduction. In this context, “reproduction” means that the parameter sets $X$ of two individuals (the “parents”) are combined to a new set. Thus, characteristics of the parameters $X$ causing a decrease of $Y$ are passed on to the next generation whereas other characteristics “die out” causing a gradual increase of the overall fitness of the population at later generations.

In order to prevent convergence of the optimisation process at local optima, a mutation is applied that randomly changes the chromosomes of 0.1 to 1% of the individuals. Defining suitable convergence criteria is not trivial for genetic optimisers because the fitness of a population may stagnate before a superior individual is found abruptly (Chipperfield et al. [33]). The optimisation process is terminated if the best individual has not improved above a problem-dependent tolerance within a certain number of generations.

In order to accelerate the optimisation process, it is iterated in three sweeps on grids
of different sizes. In sweeps 2 and 3, the best fit of the previous one is used as initial population and the allowed parameter space $[X_{\text{min}}, X_{\text{max}}]$ is decreased.

C.5.2 Parameters varied in the Matching

The parameters $X$ open for the optimiser in the matching of traverses depend on the respective field $\Phi$ to be matched. Input parameters used in the matching of total pressure and flow angles ($\Phi = [p_t, \alpha_{\text{rad}}, \alpha_{\text{tan}}]$) are coloured dark grey in Tab. C.2. Parameters open in the matching of temperature ($\Phi = T_i$) are coloured light grey. The swirl direction $d^{\text{(sw)}}$ is not changed in the matching. The remaining parameters are not open but linked to the first order swirl parameters as discussed in Appendix C.4.

<table>
<thead>
<tr>
<th>Field and Component</th>
<th>Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity</td>
<td>$S_{\text{ec}}, r_{\text{sw}}$, $d_{\text{sw}}$, $D_{\text{sw}}$</td>
</tr>
<tr>
<td>Inlet swirl</td>
<td>$\theta^{\text{(sw)}}$, $r^{\text{(sw)}}$, $\chi^{\text{(sw)}}$, $\psi^{\text{ec}}, \zeta^{\text{ec}}$</td>
</tr>
<tr>
<td>Endwall convergence</td>
<td>$u^{\text{(fe)}}_{\text{rad}}, \Delta \theta^{\text{(fe)}}, \Delta \theta^{\text{(te)}}$, $\psi^{\text{fe}}, \zeta^{\text{fe}}$</td>
</tr>
<tr>
<td>Flow entrainment</td>
<td>$u^{\text{(fe)}}_{\text{tan}}, \Delta \theta^{\text{(fe)}}, \Delta \theta^{\text{(te)}}$, $\psi^{\text{fe}}, \zeta^{\text{fe}}$</td>
</tr>
<tr>
<td>Temp.</td>
<td>$T_{\text{bg}}, r^{\text{(hs)}}, \theta^{\text{(hs)}}, \psi^{\text{bg}}, \psi^{\text{hs}}, \sigma^{\text{hs}}$</td>
</tr>
<tr>
<td>Hot spot $i$</td>
<td>$r^{\text{(hs)}}, \theta^{\text{(hs)}}, \sigma^{\text{xs},i}, \sigma^{\text{ys},i}, \phi^{\text{hs}}$</td>
</tr>
</tbody>
</table>

$\Box$ = open for matching of $\Phi = [p_t, \alpha_{\text{rad}}, \alpha_{\text{tan}}]$. $\Box$ = not open for optimiser/determined by the $\Phi_{i,0}$.

C.5.3 RRD-HPT – Computational Set-up and Mesh Study

The numerical set-up of the RRD-HPT is based on the work of Gründler [60]. A structured mesh is generated for the numerical simulations of the RRD-HPT. Information on mesh size and quality are listed in Tab. C.3. Rotor cavities are meshed using manually generated blocks that have fully matching interfaces to the main flow path, i.e., no interpolation is carried out at interfaces between cavity inlets and main flow. In the rotor tip gap, however, a non-matching interface (with interpolation) has to be used in order to prevent highly skewed cells in the rotor passage.
### Table C.3: Numerical Set-up for Simulations of the RRD-HPT

<table>
<thead>
<tr>
<th>Numerical Mesh</th>
<th>Set-up and Convergence</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain NGV1 Rotor</td>
<td>Solver ANSYS® CFX® v17.0, steady</td>
</tr>
<tr>
<td>Passages</td>
<td>Gas model ideal, compressible</td>
</tr>
<tr>
<td>Cells (Mio.)</td>
<td>Turbulence SST-k-ω, fully turbulent</td>
</tr>
<tr>
<td>Max. $y^+$</td>
<td>Avg. EVR*</td>
</tr>
<tr>
<td>Avg. $y^+$</td>
<td>Max. EVR</td>
</tr>
<tr>
<td>Avg. EVR</td>
<td>Discretisation† $\bar{\beta} \geq 0.90$ (for $u$, $P$, $h$)</td>
</tr>
<tr>
<td>Min. cell angles (°)</td>
<td>$\bar{\beta} \geq 0.47$ (for $k$, $\omega$)</td>
</tr>
<tr>
<td>Residuals</td>
<td>Imbalances $&lt; 1 \times 10^{-4}$</td>
</tr>
<tr>
<td>Imbalances $&lt; 0.0003 %$</td>
<td></td>
</tr>
</tbody>
</table>

* Element volume ratio  † Flux-blending as in Eqn. (3.13), averaged over entire domain, smallest average is reported

---

Values of non-dimensional wall distance $y^+$ exceeding 10 in the rotor cavities cannot be avoided without deteriorating mesh quality. High $y^+$ values at domain inlet are used on purpose in order to numerically dampen the boundary layer flow components in this region. Previous simulations had shown that, otherwise, oscillations in these regions lead to high solution errors and poor convergence of the overall simulation.

---

Figure C.8: Numerical set-up and mesh of the RRD-HPT
Figure C.9: Dependency of efficiency \( \eta \), NGV pressure loss \( \zeta_{\text{NGV}} \) and capacity \( \Delta \Gamma \) (left: top) and temperature distribution (left: bottom, right) on mesh size. The histograms on the right show percentage of surface area \( \delta A/A \) vs. normalised wall temperature \( T_w/T_{t,\text{in}} \) for the medium and fine mesh. A single bar (right) represents a temperature range of \( \Delta T_w = 10 \) K. Error bars (left: top) show peak-to-peak fluctuations of each variable during the last 50 iterations.

The influence of mesh resolution on results is shown in Fig. C.9. Wall temperatures are de facto grid independent on the 6.2 Mio cell mesh which is used in the analyses. Differences in efficiency \( \eta \) and capacity \( \Gamma \) between the 6.2 Mio and the 12.7 Mio cell mesh are negligible for the purpose of this work. The difference of NGV pressure loss \( \zeta_{\text{NGV}} \) is at the order of 10% of a characteristic value and therefore considerable. However, the curve for \( \zeta_{\text{NGV}} \) in Fig. C.9 implies that this difference is not only due to a reduced discretisation error but possibly also due to an increase of the solution error (Discretisation errors decrease with finer mesh resolution but the overall error is found to increase.).
C.5.4 RRD-HPT – Circumferential Averages

The matching of circumferentially averaged flow profiles, used in the evaluation of the combustor exit flow model in Section 4.10.3, at the inlet and exit of the RRD-HPT NGV row are shown in Fig. C.10 at two clocking positions.

(a) Zero clocking position ($\theta^{sw} = 0$)

(b) Half pitch clocking position ($\theta^{sw} = 0.5$ pitch)

Figure C.10: Matching of inlet traverses and NGV exit profiles
C.5.5 Matched Traverses

Tab. C.4 shows the parameter combinations used for matching the swirl field of the traverses investigated in this work.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>LSTR (SwL)</th>
<th>RRD-HPT</th>
<th>RRD-HPT (generic)</th>
<th>E3E</th>
</tr>
</thead>
<tbody>
<tr>
<td>$S$</td>
<td>0.144</td>
<td>0.235</td>
<td>0.000, 0.100, 0.200</td>
<td>0.163</td>
</tr>
<tr>
<td>$\Delta \theta^{\text{sw}}$</td>
<td>-4.613</td>
<td>10.600</td>
<td>-11.25, ..., 11.25</td>
<td>-0.484</td>
</tr>
<tr>
<td>$\Delta r^{\text{sw}}$</td>
<td>-0.167</td>
<td>-0.014</td>
<td>0.000</td>
<td>-0.024</td>
</tr>
<tr>
<td>$\chi^{\text{sw}}$</td>
<td>0.27</td>
<td>0.32</td>
<td>0.3</td>
<td>0.56</td>
</tr>
<tr>
<td>$d^{\text{sw}}$</td>
<td>-1</td>
<td>-1</td>
<td>+1/-1</td>
<td>-1</td>
</tr>
<tr>
<td>$D^{\text{sw}}$</td>
<td>0.316</td>
<td>0.332</td>
<td>0.200</td>
<td>0.118</td>
</tr>
<tr>
<td>$\alpha_{\text{rad,h}}^{\text{ec}}$</td>
<td>1.758</td>
<td>5.919</td>
<td>0.000</td>
<td>0.012</td>
</tr>
<tr>
<td>$\alpha_{\text{rad,s}}^{\text{ec}}$</td>
<td>11.582</td>
<td>3.728</td>
<td>0.000</td>
<td>23.988</td>
</tr>
<tr>
<td>$\psi_{\text{h}}^{\text{ec}}$</td>
<td>10.610</td>
<td>6.751</td>
<td>2.000</td>
<td>8.248</td>
</tr>
<tr>
<td>$\psi_{\text{s}}^{\text{ec}}$</td>
<td>27.523</td>
<td>13.544</td>
<td>2.000</td>
<td>6.280</td>
</tr>
<tr>
<td>$\zeta_{\text{h}}^{\text{ec}}$</td>
<td>0.516</td>
<td>0.791</td>
<td>0.800</td>
<td>0.600</td>
</tr>
<tr>
<td>$\zeta_{\text{s}}^{\text{ec}}$</td>
<td>3.525</td>
<td>0.848</td>
<td>0.800</td>
<td>5.474</td>
</tr>
<tr>
<td>$u_{\text{rad,h}}^{\text{fe}}$</td>
<td>m/s</td>
<td>3 max[$u_{\text{rad}}^{\text{sw}}$] (→ Tab. C.1)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$u_{\text{rad,s}}^{\text{fe}}$</td>
<td>m/s</td>
<td>2 max[$u_{\text{rad}}^{\text{sw}}$] (→ Tab. C.1)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\Delta \theta_{\text{rad,h}}^{\text{fe}}$</td>
<td>-3.649</td>
<td>1.826</td>
<td>0.000</td>
<td>-4.984</td>
</tr>
<tr>
<td>$\Delta \theta_{\text{rad,s}}^{\text{fe}}$</td>
<td>3.337</td>
<td>-1.545</td>
<td>0.000</td>
<td>11.329</td>
</tr>
<tr>
<td>$\chi_{\text{rad,h}}^{\text{fe}}$</td>
<td>$r^{\text{sw}} - 0.5$ (→ Tab. C.1)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\chi_{\text{rad,s}}^{\text{fe}}$</td>
<td>$r^{\text{sw}} + 0.5$ (→ Tab. C.1)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\psi_{\text{rad,h}}^{\text{fe}}$</td>
<td>0.230</td>
<td>0.817</td>
<td>1.000</td>
<td>5.935</td>
</tr>
<tr>
<td>$\psi_{\text{rad,s}}^{\text{fe}}$</td>
<td>3.125</td>
<td>5.112</td>
<td>1.000</td>
<td>7.129</td>
</tr>
<tr>
<td>$u_{\text{tan,h}}^{\text{fe}}$</td>
<td>m/s</td>
<td>Determined from $u_{\text{tan}}(\theta_{sb}, r_h) = 0$ (→ Tab. C.1)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$u_{\text{tan,s}}^{\text{fe}}$</td>
<td>m/s</td>
<td>Determined from $u_{\text{tan}}(\theta_{sb}, r_s) = 0$ (→ Tab. C.1)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\Delta \theta_{\text{tan,h}}^{\text{fe}}$</td>
<td>6.234</td>
<td>3.256</td>
<td>0.000</td>
<td>6.078</td>
</tr>
<tr>
<td>$\Delta \theta_{\text{tan,s}}^{\text{fe}}$</td>
<td>-1.355</td>
<td>0.936</td>
<td>0.000</td>
<td>-3.331</td>
</tr>
<tr>
<td>$\psi_{\text{tan,h}}^{\text{fe}}$</td>
<td>0.180</td>
<td>1.579</td>
<td>1.000</td>
<td>0.111</td>
</tr>
<tr>
<td>$\psi_{\text{tan,s}}^{\text{fe}}$</td>
<td>0.976</td>
<td>0.110</td>
<td>1.000</td>
<td>0.440</td>
</tr>
</tbody>
</table>

C.5 Evaluation of Combustor Exit Flow Model
C.5.6 RRD-HPT – Additional Test Cases of Matched Inlet Conditions

RRD-HPT – Test Case 1

<table>
<thead>
<tr>
<th>Clocking \ /pitch</th>
<th>Hub</th>
<th>Pressure Side</th>
<th>Suction Side</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Datum</td>
<td>Match</td>
<td>Datum</td>
</tr>
<tr>
<td>0</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>1/8</td>
<td><img src="image5" alt="Image" /></td>
<td><img src="image6" alt="Image" /></td>
<td><img src="image7" alt="Image" /></td>
</tr>
<tr>
<td>2/8</td>
<td><img src="image9" alt="Image" /></td>
<td><img src="image10" alt="Image" /></td>
<td><img src="image11" alt="Image" /></td>
</tr>
<tr>
<td>3/8</td>
<td><img src="image13" alt="Image" /></td>
<td><img src="image14" alt="Image" /></td>
<td><img src="image15" alt="Image" /></td>
</tr>
<tr>
<td>4/8</td>
<td><img src="image17" alt="Image" /></td>
<td><img src="image18" alt="Image" /></td>
<td><img src="image19" alt="Image" /></td>
</tr>
<tr>
<td>5/8</td>
<td><img src="image21" alt="Image" /></td>
<td><img src="image22" alt="Image" /></td>
<td><img src="image23" alt="Image" /></td>
</tr>
<tr>
<td>6/8</td>
<td><img src="image25" alt="Image" /></td>
<td><img src="image26" alt="Image" /></td>
<td><img src="image27" alt="Image" /></td>
</tr>
<tr>
<td>7/8</td>
<td><img src="image29" alt="Image" /></td>
<td><img src="image30" alt="Image" /></td>
<td><img src="image31" alt="Image" /></td>
</tr>
</tbody>
</table>

Figure C.11: Matching of test case 1
RRD-HPT – Test Case 2

<table>
<thead>
<tr>
<th>Clocking /pitch</th>
<th>Hub Datum</th>
<th>Hub Match</th>
<th>Pressure Side Datum</th>
<th>Pressure Side Match</th>
<th>Suction Side Datum</th>
<th>Suction Side Match</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>1/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>2/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>4/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>5/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>6/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>7/8</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Figure C.12: Matching of test case 2
RRD-HPT – Test Case 3

<table>
<thead>
<tr>
<th>Clocking / pitch</th>
<th>Hub Datum Match</th>
<th>Pressure Side Datum Match</th>
<th>Suction Side Datum Match</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
</tr>
<tr>
<td>1/8</td>
<td><img src="image4" alt="Image" /></td>
<td><img src="image5" alt="Image" /></td>
<td><img src="image6" alt="Image" /></td>
</tr>
<tr>
<td>2/8</td>
<td><img src="image7" alt="Image" /></td>
<td><img src="image8" alt="Image" /></td>
<td><img src="image9" alt="Image" /></td>
</tr>
<tr>
<td>3/8</td>
<td><img src="image10" alt="Image" /></td>
<td><img src="image11" alt="Image" /></td>
<td><img src="image12" alt="Image" /></td>
</tr>
<tr>
<td>4/8</td>
<td><img src="image13" alt="Image" /></td>
<td><img src="image14" alt="Image" /></td>
<td><img src="image15" alt="Image" /></td>
</tr>
<tr>
<td>5/8</td>
<td><img src="image16" alt="Image" /></td>
<td><img src="image17" alt="Image" /></td>
<td><img src="image18" alt="Image" /></td>
</tr>
<tr>
<td>6/8</td>
<td><img src="image19" alt="Image" /></td>
<td><img src="image20" alt="Image" /></td>
<td><img src="image21" alt="Image" /></td>
</tr>
<tr>
<td>7/8</td>
<td><img src="image22" alt="Image" /></td>
<td><img src="image23" alt="Image" /></td>
<td><img src="image24" alt="Image" /></td>
</tr>
</tbody>
</table>

\[ \Delta \theta_{sw} \] (pitch)

\[ T_{w,min} \] – \[ T_{w,max} \]

**Figure C.13:** Matching of test case 3
### RRD-HPT – Test Case 4

<table>
<thead>
<tr>
<th>Clocking /pitch</th>
<th>Hub</th>
<th>Pressure Side</th>
<th>Suction Side</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Datum</td>
<td>Match</td>
<td>Datum</td>
</tr>
<tr>
<td>0</td>
<td>![Image]</td>
<td>![Image]</td>
<td>![Image]</td>
</tr>
</tbody>
</table>

**Figure C.14:** Matching of test case 4
RRD-HPT – Test Case 5

<table>
<thead>
<tr>
<th>Clocking /pitch</th>
<th>Hub</th>
<th>Pressure Side</th>
<th>Suction Side</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Datum Match</td>
<td>Datum Match</td>
<td>Datum Match</td>
</tr>
<tr>
<td>0</td>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
<tr>
<td>1/8</td>
<td><img src="image4.png" alt="Image" /></td>
<td><img src="image5.png" alt="Image" /></td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
<tr>
<td>2/8</td>
<td><img src="image7.png" alt="Image" /></td>
<td><img src="image8.png" alt="Image" /></td>
<td><img src="image9.png" alt="Image" /></td>
</tr>
<tr>
<td>3/8</td>
<td><img src="image10.png" alt="Image" /></td>
<td><img src="image11.png" alt="Image" /></td>
<td><img src="image12.png" alt="Image" /></td>
</tr>
<tr>
<td>4/8</td>
<td><img src="image13.png" alt="Image" /></td>
<td><img src="image14.png" alt="Image" /></td>
<td><img src="image15.png" alt="Image" /></td>
</tr>
<tr>
<td>5/8</td>
<td><img src="image16.png" alt="Image" /></td>
<td><img src="image17.png" alt="Image" /></td>
<td><img src="image18.png" alt="Image" /></td>
</tr>
<tr>
<td>6/8</td>
<td><img src="image19.png" alt="Image" /></td>
<td><img src="image20.png" alt="Image" /></td>
<td><img src="image21.png" alt="Image" /></td>
</tr>
<tr>
<td>7/8</td>
<td><img src="image22.png" alt="Image" /></td>
<td><img src="image23.png" alt="Image" /></td>
<td><img src="image24.png" alt="Image" /></td>
</tr>
</tbody>
</table>

\[ T_{w,\text{min}} \quad \text{to} \quad T_{w,\text{max}} \]

**Figure C.15:** Matching of test case 5
C.5.7 RRD-HPT – Match of Wall Temperatures and Heat Transfer

The histograms in Fig. C.16 and C.17 enable a quantitative comparison of the distribution of Nu and $T_w$, respectively, over area fraction $\delta A/A$. It is shown which percentage of total area $\delta A/A$ (on the ordinate) has a respective value of Nu or $T_w$ (on the abscissa).

**Figure C.16:** Comparison of Nusselt number Nu at the turbine walls using datum inlet condition from combustor CFD and the parameterised, matched inlet condition at four different clocking positions. Distributions are shown as histograms of area fraction $\delta A/A$ vs. Nu where each bar corresponds to a range of $\Delta Nu = 10$. The reference temperature $T_{ref}$ was set constant at approx. 60% of $T_{t, in}$.

**Figure C.17:** Comparison of normalised adiabatic wall temperature $T_w/T_{t, in}$ using datum inlet condition from combustor CFD and the matched, parameterised inlet condition at four different clocking positions. Distributions are shown as histograms of area fraction $\delta A/A$ vs. $T_w/T_{t, in}$ where each bar corresponds to a temperature range of $\Delta T_w = 10$ K. Red bars display the averaged values $T_w/T_{t, in}$.
C.5.8 RRD-HPT – Inlet Velocity Correction

As pointed out in Section 4.4, knowledge of the distribution of axial velocity \( u_{ax}(r, \theta) \) is required in order to scale inlet stagnation temperature to operating conditions. However, \( u_{ax}(r, \theta) \) cannot be provided from information available in the traverse model. In this work, \( u_{ax}(r, \theta) \) is therefore guessed (either from a known reference traverse or as a homogeneous profile) in order to generate an initial traverse which is applied as inlet condition to a dummy CFD run. The distribution of \( u_{ax}^*(r, \theta) \) from this simulation is then fed back to the model to generate a corrected traverse. The unknown static pressure profile is treated equally. Applying this corrected traverse to a second CFD simulation will yield another distribution of axial velocity which is not necessarily equal to \( u_{ax}^*(r, \theta) \). Therefore, the correction can be repeated using \( u_{ax}^*(r, \theta) \) as input. If this procedure would be repeated ad infinitum, one would expect \( u_{ax}^*(r, \theta) \) to converge to a stable state after a certain number of iterations.

This presumption is proved wrong by the following investigation: A generic inlet traverse \( \Phi_{BC} \) with inlet swirl and a hot spot (Fig. C.18, top left) is generated from the initial assumption \( u_{ax,0}(r, \theta) = \text{const.} \) which is then iteratively corrected using distributions \( u_{ax}^*(r, \theta) \) from nine successive CFD runs of the RRD-HPT set-up (Appendix C.5.3). Similarly, the pressure field \( p(r, \theta) \) is initially assumed constant and then updated using the CFD distributions \( p^*(r, \theta) \).

In Fig. C.18, differences in mass flow averaged axial velocity \( \overline{u_{ax}}(r, \theta) \) to the conditions specified in the model are shown as grey bars for the different iterations of the correction process. It can be seen that \( \overline{u_{ax}}(r, \theta) \) does not converge towards a stable state but rather grows boundless with progression of the correction process. Coloured contours show the 2D distribution of axial velocity as well as the difference to the distribution from the
previous iteration. It can be seen that in the region of flow entrainment from the hub local differences in $u_{ax}^*(r, \theta)$ persist between successive iterations (the difference between iterations 4 and 5 persists indefinitely) which causes the drift in $u_{ax}(r, \theta)$ and, as a consequence, a similar drift in mean inlet temperature $T_{i,in}$. This behaviour is due to the coupling of both quantities via the density field and can be observed independent of the initial guess of $u_{ax,0}(r, \theta)$. It can also be seen that significant changes in velocity distribution, with local differences at the order of 50\% in $u_{ax}(r, \theta)$, occur only in the first iteration. From the second iteration on, the topology of $u_{ax}(r, \theta)$ remains relatively constant. It is therefore concluded that one iteration in the correction of axial velocity is sufficient in order to adjust the topology of $u_{ax}(r, \theta)$, used in the model, to the one that will result in CFD and keep the drift in $u_{ax}(r, \theta)$ within acceptable bounds.

It is emphasised that the described correction process can only provide a consistent traverse in the sense that the assumed velocity field matches the initially generated total pressure field. It cannot correct for errors in the initial guess for $u_{ax}(r, \theta)$ and $p(r, \theta)$ because its error propagates to the next iteration via the total pressure distribution (cf. Section 4.5). The initial guess for $u_{ax}(r, \theta)$ should therefore be as close to the “real” traverse as possible, i.e., retrieved from a converged CFD simulation in order to minimise errors.

---

1 The same behaviour is observed for initial traverses, derived from converged CFD, using the same inlet traverse without homogeneous inlet temperature and different swirl clocking positions.
D Appendix to Results

D.1 RRD-HPT – Losses and Vane Secondary Flows Induced by Inlet Swirl

D.1.1 RRD-HPT – Efficiency Variation at Pressure Outlet Condition

The results shown in Fig. 5.2 are produced using a mass flow outlet condition, thus prescribing a constant capacity $\Gamma$. The study is repeated with a fixed outlet pressure in order to allow for a variation in $\Gamma$. Results from this study are shown in Fig. D.1.

![Figure D.1: Efficiency deficit at increasing swirl number: The colour of marker indicates $\Gamma$, the colour of marker edges indicates $\Delta \theta^{(sw)}$.](image)

D.1.2 RRD-HPT – Comparison of SST and RSM Turbulence Modelling

The inability of two-equation models to correctly resolve streamline curvature is discussed in Section 3.1.3. Simulations of the RRD-HPT are therefore run on two different set-ups in order to assess effects of turbulence modelling on the findings: A stage set-up (including one stator and one rotor row) using the SST-$k$-$\omega$ model is used as a baseline which is compared with a second set-up containing only the NGV1 row (enabling stability of the CFD) using the RSM-BSL model. The circumferentially averaged static pressure from the stage calculations is used as outlet condition of the single row calculations.

The NGV exit pressure profiles shown in Section 5.2.3 are computed by means of two-equation turbulence modelling. In order to show that the qualitative results drawn from these profiles are independent of turbulence modelling, the analysis is repeated at certain clocking positions using the SSG-Reynolds Stress model. A comparison between SST and SSG turbulence modelling is shown in Fig. D.2.
Results from the SST-$k$-$\omega$ turbulence model are shown in the top and bottom row for $S = 0.2$ and $S = 0.1$, respectively. Single row computations using the RSM-BSL Reynolds stress turbulence model at $S = 0.2$ are shown in the centre row. The zones \( A \) to \( D \) can still be observed in the profiles with the same trends in position and size. The largest differences can be observed in zone \( B \) which shows two separate peaks with the SSG model.

![Diagram showing total pressure loss profiles](image)

**Figure D.2:** Total pressure loss profiles at NGV1 exit for different clocking positions of inlet swirl compared with an axial reference at positive swirl rotation

Pressure loss coefficient $\eta_{\text{NGV}}$ is compared for the two set-ups in Fig. D.3. The single-row simulations using the RSM (grey curve) qualitatively reproduce the trends in
NGV pressure loss. A quantitative difference, however, is visible due to different operating conditions imposed by the outlet BCs. The nozzle is not choked and converges to a different mass flow in the single row configuration. Note, that the solution error level of the single-row calculations is an order of magnitude higher than that of the stage calculations.

Figure D.3: Influence of turbulence modelling on NGV pressure loss

D.1.3 Influence of Transient Blade Row Interaction

The influence of transient blade row interaction on the results shown in Fig. 5.3 is investigated by comparing the steady mixing-plane simulations with unsteady simulations using the time transformation approach (cf. Section B.2). The comparison in Tab. D.1 shows that TBR causes difference in efficiency\(^1\) of \(\Delta \eta = -0.17\%\) with axial inflow. This difference is only slightly smaller in case of inlet swirl at the investigated clocking positions with \(-0.14\%\) and \(-0.11\%\), respectively. The results shown in Fig. 5.3 thus deviate by about 5\% from predictions taking TBR into account.

Table D.1: Efficiency Difference with respect to a Steady Simulation with Axial Inflow

<table>
<thead>
<tr>
<th>Swirl number ( S )</th>
<th>0.0</th>
<th>0.2</th>
<th>0.2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Clocking ( \Delta \theta_{{sw}} ) (pitch)</td>
<td>-1/2</td>
<td>-1/4</td>
<td></td>
</tr>
<tr>
<td>Steady ( \Delta \eta ) (%)</td>
<td>-0.56</td>
<td>-0.90</td>
<td></td>
</tr>
<tr>
<td>Unsteady ( \Delta \eta ) (%)</td>
<td>-0.17 ± 0.10</td>
<td>-0.70 ± 0.01</td>
<td>-1.01 ± 0.07</td>
</tr>
</tbody>
</table>

\(^1\) Efficiency differences are reported as mean ± half amplitude of efficiency fluctuations of the last 20 rotor-stator blade passages.
D.1.4 RRD-HPT – Efficiency Loss Correlation

Efficiency loss from inlet swirl $\Delta \eta$ at different swirl strengths $S$ and clocking positions $\Delta \theta^{\text{sw}}$ can be approximated by the function

$$\Delta \eta/\% \approx A \sin \left( C_3 \frac{\Delta \theta^{\text{sw}}}{\theta_{\text{pitch}}} + B \right) + C,$$

(A.1)

with the parameters $C_1$ to $C_5$. An approximation of the data from Fig. 5.3 with the corresponding set of parameters are shown in Fig. D.4. The maximal deviation between the data points and the approximation is $\Delta \eta \approx 0.1 \%$. Swirl clocking $\theta^{\text{sw}}$ is referenced to the zero position shown in Fig. 5.1. For the investigated case, $S$ is related to maximal swirl angles in the traverse $\alpha_{\text{tan}}$ as

$$S(\alpha_{\text{tan}}) = 1.32 \times 10^{-4} \alpha_{\text{tan}}^2 + 1.675 \times 10^{-4} \alpha_{\text{tan}},$$

(D.2)

<table>
<thead>
<tr>
<th>$C_i$</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$C_1$</td>
<td>0.01512</td>
</tr>
<tr>
<td>$C_2$</td>
<td>15.27</td>
</tr>
<tr>
<td>$C_3$</td>
<td>6.29</td>
</tr>
<tr>
<td>$C_4$</td>
<td>3.0</td>
</tr>
<tr>
<td>$C_5$</td>
<td>0.3</td>
</tr>
<tr>
<td>$C_6$</td>
<td>-0.0206</td>
</tr>
<tr>
<td>$C_7$</td>
<td>17.11</td>
</tr>
</tbody>
</table>

**Figure D.4: Approximation of efficiency loss from inlet swirl**
D.2 LSTR – Set-up of Swirler Computations

Figure D.5: The domain used for numerical simulations of the LSTR swirl generator shown schematically in a cut through the measurement section of the test rig (left) and as a 3D model (right)

The numerical model used for simulations of the LSTR swirl generator, referred to in Section 5.3.2, is shown in Fig. D.5. The model was set-up by Bäumler [13]. Film cooling is not resolved in the numerical model and the vanes are simplified as prismatic extrusions of the midspan profile in order to reduce the size of the numerical mesh which is strongly refined in the recirculation zone behind the swirler. Due to the high effort associated with unsteady simulations, the size of the final mesh is determined in a mesh study assuming steady-state (not shown).

Table D.2: Numerical Set-up for Simulations of the LSTR Swirl Generator

<table>
<thead>
<tr>
<th>Numerical Mesh</th>
<th>Set-up and Convergence</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cells (Mio.)</td>
<td>23.7</td>
</tr>
<tr>
<td>Max. EVR*</td>
<td>139.52</td>
</tr>
<tr>
<td>Avg. EVR</td>
<td>2.48</td>
</tr>
<tr>
<td>Min. cell angles (°)</td>
<td>21.2</td>
</tr>
<tr>
<td>Solver</td>
<td>ANSYS® CFX® v17.0, unsteady</td>
</tr>
<tr>
<td>Time step</td>
<td>$\Delta t = 1 \times 10^{-4}$ s</td>
</tr>
<tr>
<td>Gas model</td>
<td>incompressible, isothermal</td>
</tr>
<tr>
<td>Turbulence</td>
<td>RSM-SSG, wall functions</td>
</tr>
<tr>
<td>Discretisation†</td>
<td>$\bar{\beta} \geq 0.9$ for $(u, P)$</td>
</tr>
<tr>
<td></td>
<td>$\bar{\beta} \geq 0.65$ for $(\bar{u}_i \bar{u}_j, \omega)$</td>
</tr>
<tr>
<td></td>
<td>2. order implicit Euler in time</td>
</tr>
<tr>
<td>Residuals</td>
<td>$&lt; 5 \times 10^{-5}$</td>
</tr>
<tr>
<td>Imbalances</td>
<td>$&lt; 0.005$ %</td>
</tr>
</tbody>
</table>

* Element volume ratio
† Flux-blending as in Eqn. (3.13), averaged over entire domain, smallest average is reported
D.3 LSTR – Influence of Total Pressure Profile

As discussed in Section 4.5, there is a certain error in the inlet total pressure profile computed in the flow model. In Section 5.3.2 it is shown that the inlet total pressure profile has an effect on the quantitative results of the UQ analysis in the LSTR NGV row. A comparison of the results in Fig. D.6, obtained using the same datum inlet total pressure profile in all simulations, with those in Fig. 5.13, obtained with the inlet total pressure profile predicted by the flow model, shows that qualitative results with respect to film cooling effectiveness $\eta_{FC}$ are independent of the inlet total pressure profile.

**Mean and Standard Deviation** Film Cooling Effectiveness

![Heatmap showing mean and standard deviation of film cooling effectiveness](image)

**Correlation Coefficients** Film Cooling Effectiveness

![Heatmap showing correlation coefficients](image)

**Figure D.6:** Mean $\mu$ and standard deviation $\sigma$ of film cooling effectiveness on the vanes (top, computed from the gPC expansion $\tilde{Y}$) and correlations $\rho$ with the parameters determining the inlet traverse (bottom, computed from the Monte Carlo samples $\tilde{Y}$) with the reference inlet total pressure profile used in all simulations. Correlations are shown quantitatively in regions of high deviations ($\sigma(\eta_{FC}) = 1.5\%$) only for better visibility.
D.4 LSTR – Correlations with Heat Transfer

Correlations of the input parameters $X$ to the inlet traverse with film cooling effectiveness $\eta_{FC}$ on the vanes of the LSTR NGV1 are shown in Fig. 5.13. In addition, correlations with Nusselt number $\text{Nu}$ are shown in Fig. D.7.

**Figure D.7:** Correlations $\rho$ with the parameters determining the inlet traverse (computed from the Monte Carlo samples $\tilde{Y}$.) with Nusselt number. Correlations are shown quantitatively in regions of high deviations ($\sigma(\text{Nu}) = 300$) only for better visibility.
D.5 E3E – Influence of Turbulence Modelling

D.5.1 E3E – Influence on Swirl-Induced Secondary Flows

Fig. D.9 provides a larger version of the comparison of the prediction of the induced vortex at the LE of the E3E NGV1 with different turbulence models shown in Fig. 5.17. Simulations are conducted on the stator row only. Streamlines are visualised on a plane, which is approximately aligned with the stagnation line of the first vane. It can be seen that the flow features are predicted qualitatively equally by both turbulence models.

![Figure D.8: Induced swirl structure at Engine 3E NGV LE](image1)

D.5.2 E3E – Influence on NGV1 Wall Temperatures

The influence of turbulence modelling on wall temperature distribution in the E3E NGV1 row shown in Fig. D.9. Note, that wall functions were applied in the computations using the RSM-SSG model.

![Figure D.9: Histograms of adiabatic wall temperature in the E3E NGV1 row obtained by means of different turbulence models](image2)
D.6 E3E – Nusselt Number for Datum and Matched Inlet Conditions

Figure D.10: Comparison of Nusselt number at the hub and vanes of the E3E for datum and matched 2D inlet conditions: Distributions from homogeneous axial inflow are shown as reference.

Distribution of Nu on the hub end wall and vanes of the E3E first stator row is shown in Fig. D.10 for the datum 2D inlet conditions, the matched inlet conditions
and an reference with axial, homogeneous inflow. The maxima of $Nu \approx 2500$ on the SS shoulder agree qualitatively and quantitatively with the results of Schmid [157].

D.7 Deriving Uncertainties of the Hot Streak Position from Test Data

![Diagram showing Combustor Geometry A and B with measured and fitted data](image)

**Figure D.11:** Derivation of uncertainties of the parameters determining the hot spot behind two real engine combustors by recreating measured traverses using the traverse model and fitting Normal distributions to the data.

Characteristic deviations of the hot spot are derived by matching the distribution of measured temperature traverses at different sectors along the circumference behind two real combustion chambers using the traverse model with a single hot spot (Fig.
Standard deviations of the different parameters are then determined by matching Normal distributions to the data as shown in the histograms.

### D.8 E3E – Results of Uncertainty Quantification Studies

#### D.8.1 E3E – Uncertainty Propagation

Deviations of operating conditions are reported in terms of capacity $\Gamma$ and mean inlet temperature $T_{t, in}$. Variations in $p_{t, in}$ are below 0.02% and therefore not reported. Results for each run are shown in three rows of which the first represents the deviations of the raw data obtained by Monte Carlo evaluation, the second row represents the values obtained after correction for inlet temperature deviations by Eqn. (5.2) and the third row gives values obtained from gPC expansion of the corrected data. It can be seen that results from crude MCS and gPC expansions are in close agreement. As described in Section 5.4.1, the first run a) is repeated on a finer mesh in order to evaluate the influence of mesh resolution on the results. The respective values are shown in a fourth row for this run.

<table>
<thead>
<tr>
<th>Standard deviations of inputs $X$</th>
<th>Mean ± standard deviation of outputs $Y$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_{t, in}$ max (%)</td>
<td>$\eta$ (%)</td>
</tr>
<tr>
<td>r (%)</td>
<td>$\theta_{hs}$ (°)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Run</th>
<th>$\Delta T_{t, in}$ (%)&lt;sup&gt;†&lt;/sup&gt;</th>
<th>$\Delta \theta_{hs}$ (°)&lt;sup&gt;†&lt;/sup&gt;</th>
<th>$\Delta S$ (-)</th>
<th>$\Delta \theta_{sw}$ (°)&lt;sup&gt;†&lt;/sup&gt;</th>
<th>$\eta$ (%)&lt;sup&gt;§&lt;/sup&gt;</th>
<th>$\zeta_{NGV}$ (%)&lt;sup&gt;§&lt;/sup&gt;</th>
<th>$\Gamma$ (%)&lt;sup&gt;§&lt;/sup&gt;</th>
<th>$T_{t, in}$ (%)&lt;sup&gt;§&lt;/sup&gt;</th>
</tr>
</thead>
<tbody>
<tr>
<td>a)</td>
<td>-</td>
<td>-</td>
<td>0.025</td>
<td>1.5</td>
<td>-0.14±0.08</td>
<td>9.95±0.08</td>
<td>-0.06±0.05</td>
<td>0.46±0.09</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.00±0.08</td>
<td>9.93±0.08</td>
<td>-0.04±0.05</td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.01±0.09</td>
<td>9.93±0.10</td>
<td>-0.04±0.06</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.22±0.05</td>
<td>9.79±0.09</td>
<td>-0.05±0.05</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>N = 9 ($n_p = 1.5$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>b)</td>
<td>-</td>
<td>-</td>
<td>0.050</td>
<td>1.5</td>
<td>-0.13±0.16</td>
<td>9.97±0.18</td>
<td>-0.07±0.11</td>
<td>0.45±0.12</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>0.01±0.14</td>
<td>9.94±0.17</td>
<td>-0.05±0.11</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>N = 20 ($n_p = 3.3$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>c)</td>
<td>2.5</td>
<td>10</td>
<td>1.5</td>
<td>-</td>
<td>-0.08±0.08</td>
<td>9.89±0.04</td>
<td>-0.03±0.01</td>
<td>-0.53±0.16</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.25±0.13</td>
<td>9.92±0.04</td>
<td>-0.06±0.01</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>N = 20 ($n_p = 2.0$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>d)</td>
<td>2.5</td>
<td>10</td>
<td>1.5</td>
<td>0.025</td>
<td>1.5</td>
<td>-0.08±0.10</td>
<td>9.88±0.09</td>
<td>-0.03±0.05</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>-0.24±0.14</td>
<td>9.91±0.09</td>
<td>-0.05±0.05</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>N = 21 ($n_p = 1.0$)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

\* % of $T_{t, in}$; \ † % of span height; \ ‡ % of datum conditions; \ § = computed using gPC; \ = computed on finer mesh (76.4 Mio. cells)

#### D.8.2 E3E – Correlation Matrices

The influence of temperature correction, Eqn. (5.2), and the statistical method used (crude Monte Carlo sampling or gPC) on the computed correlations can be assessed.
by comparing the correlation matrices in Fig. D.12 and Fig. D.13 with those shown in Fig. 5.22. Temperature correction affects mostly the correlations with inlet mass flow \( \dot{m}_{in} \). There are quantitative differences between correlations computed from the raw MCS data and the gPC expansions but the qualitative conclusions can be drawn equally from the results of both methods.

### Figure D.12: Correlation matrix obtained using gPC expansions without temperature correction

### Figure D.13: Correlation matrices obtained using the (temperature corrected) Monte Carlo sampled raw data

#### D.8.3 E3E – Relation of Film Cooling Effectiveness and Wall Temperature

Adiabatic wall temperature \( T_{w,ad} \) and adiabatic film cooling effectiveness are related by

\[
T_{w,ad} = T_{\infty} - \eta_{FC}(T_{\infty} - T_c).
\]

If free stream temperature \( T_{\infty} \) and coolant temperature \( T_c \) remain unchanged between two configurations, a difference in adiabatic wall temperature \( \Delta T_{w,ad} = \)
$T_{\text{w,ad}} - T'_{\text{w,ad}}$ is proportional to a difference in film cooling effectiveness $\Delta \eta_{\text{FC}} = \eta_{\text{FC}} - \eta'_{\text{FC}}$ because

$$
\Delta T_{\text{w,ad}} = T_\infty - \eta_{\text{FC}}(T_\infty - T_c) - [T'_\infty - \eta'_{\text{FC}}(T'_\infty - T'_c)] \\
= -(\eta_{\text{FC}} - \eta'_{\text{FC}})(T_\infty - T_c) \\
= \Delta \eta_{\text{FC}}(T_c - T_\infty).
$$

(D.4)

If the HPT inlet temperature is used as free stream temperature, $T_\infty = \overline{T_{\text{t,in}}}$, the proportionality can be expressed as

$$
\frac{\Delta T_{\text{w,ad}}}{\overline{T_{\text{t,in}}}} = \left( \frac{\overline{T_c}}{\overline{T_{\text{t,in}}}} - 1 \right) \Delta \eta_{\text{FC}} = -0.56 \Delta \eta_{\text{FC}},
$$

(D.5)

where $T_c/\overline{T_{\text{t,in}}} = 0.44$ is set in accordance with the conducted CFD simulations of E3E.

D.8.4 E3E – Temperature-Weighted Standard Deviation of Wall Temperature

With regard to practical design problems, the uncertainties in adiabatic wall temperature shown in Fig. 5.25 may not be directly relevant but one might rather be interested particularly in regions at the wall that have a high temperature and a high uncertainty at the same time. In order to highlight these regions, a temperature weighted standard deviation of wall temperature is defined by

$$
\sigma' \left( \frac{T_{\text{w,ad}}}{\overline{T_{\text{t,in}}}} \right) := \mu \left( \frac{T_{\text{w,ad}}}{\overline{T_{\text{t,in}}}} - \overline{T_c} \right) \times \sigma \left( \frac{T_{\text{w,ad}}}{\overline{T_{\text{t,in}}}} \right)
$$

(D.6)

and compared to the uniformly weighted standard deviation in Fig. D.14. The regions of high temperature in Fig. 5.19 show higher values of $\sigma'(T_{\text{w,ad}})$ at the PS of the central and right vane but both formulations of $\sigma$ yield equal qualitative results, especially at the TE of the left vane.

Figure D.14: Comparison of standard deviation of wall temperature $\sigma(T_{\text{w,ad}})$ (left) and temperature weighted standard deviation $\sigma'(T_{\text{w,ad}})$ (right).
Qualitative differences in results between a similar deviation $\sigma'(\text{Nu})$ and $\sigma(\text{Nu})$ in Fig. 5.25 are smaller than those of between $\sigma'(T_{w,ad})$ and $\sigma(T_{w,ad})$ due to the more homogeneous distribution of Nu (Fig. D.10).

D.8.5 E3E – Vane Wall Temperature Correlation Coefficients

Mean and standard deviation of wall temperatures $T_w$ on the Engine 3E NGV1 due to the uncertain input parameters $X$ to the inlet traverse are shown in Fig. 5.24. In addition, correlations of the parameters $X$ with $T_w$ are shown in Fig. D.15.

![Correlation diagrams](image)

Figure D.15: Correlations $\rho$ of the parameters determining the inlet traverse in gPC run d) with adiabatic wall temperature $T_{w,ad}$ (computed from the Monte Carlo samples $\tilde{Y}$). Correlations are shown quantitatively in regions of high deviations ($\sigma(T_{w,ad}/T_{t,in}) > 5\%$) in run d) only for better visibility. The data are not corrected for deviations in mean inlet temperature $\overline{T_{t,in}}$. 

---

198 D Appendix to Results
Mean and standard deviation of Nusselt number $\text{Nu}$ on the Engine 3E NGV1 due to the uncertain input parameters $\mathbf{X}$ to the inlet traverse are shown in Fig. 5.24. In addition, correlations of the parameters $\mathbf{X}$ with $\text{Nu}$ are shown in Fig. D.16.

Figure D.16: Correlations $\rho$ of the parameters determining the inlet traverse in gPC run d) with Nusselt number $\text{Nu}$ (computed from the Monte Carlo samples $\tilde{\mathbf{Y}}$). Correlations are shown quantitatively in regions of high deviations ($\sigma(\text{Nu}) > 150$) in run d) only for better visibility. The data are not corrected for deviations in mean inlet temperature $\overline{T_{\text{t,in}}}$.
**E Appendix to Conclusion**

**E.1 Global Impact of Uncertainty in High Pressure Turbine Efficiency**

The effects of deviations in HPT efficiency $\sigma(\eta_{HPT})$ and turbine inlet temperature $\sigma(T_{t,40})$ on mean engine efficiency $\mu(\eta_o)$ and its standard deviation $\sigma(\eta_o)$ are compared using the Monte Carlo analysis of an engine cycle presented in Section 3.5.5. In order to evaluate the influence of different levels of uncertainties in $T_{t,40}$ and $\eta_{HPT}$, 30 Monte Carlo simulations with different standard deviations of the inputs

$$X = [T_{t,40}, \eta_{HPT}]$$  \hspace{1cm} (E.1)

are run, each using $N = 1000$ samples. The mean values $\mu$ and standard deviations $\sigma$ of thermal efficiency $\eta_{th}$, propulsive efficiency $\eta_{pr}$, overall efficiency $\eta_o$ and normalised core engine thrust $F_{n,core}/\dot{m}_\infty a_\infty$ are evaluated for each of the 30 simulations,

$$Y = [\eta_{th}, \eta_{pr}, \eta_o, \frac{F_{n,core}}{\dot{m}_\infty a_\infty}].$$  \hspace{1cm} (E.2)

The mean values and deviations of $Y$ from each of the 30 simulations are arranged in a $5 \times 6$ grid and interpolated. Iso-lines of this interpolation are shown in Fig. E.1 and allow for a comparison of the influence of the relative uncertainties in $T_{t,40}$ and $\eta_{HPT}$ on the results $Y$. The datum values corresponding to the cycle in Fig. B.1 are shown in Tab. E.1.

<table>
<thead>
<tr>
<th>Uncertain Inputs X</th>
<th>Uncertain Outputs Y</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T_{t,40}$</td>
<td>$\eta_{HPT}$</td>
</tr>
<tr>
<td>1624.5 K</td>
<td>90.0 %</td>
</tr>
</tbody>
</table>

It can be seen that $\sigma(\eta_{HPT})$ only significantly affects the uncertainty of the engine’s thermal efficiency $\sigma(\eta_{th})$ and thus also $\sigma(\eta_o)$. Propulsive efficiency and thrust are almost unaffected by $\sigma(\eta_{HPT})$. A maximal uncertainty in $\eta_{HPT}$ of about 0.2 % for a fixed $T_{t,40}$ (cf. Tab. 5.4) corresponds to an uncertainty in $\eta_o$ of approximately 0.05 %. However, it can be seen that with increasing uncertainty of $T_{t,40}$ the influence of $\sigma(\eta_{HPT})$ on $\sigma(\eta_{th})$ decreases rapidly. Therefore, typical uncertainties in $T_{t,40}$ well
above 1% (cf. Fig. 1.4) are likely to overshadow the effects of $\sigma(\eta_{\text{HPT}})$ and dominate the uncertainty in $\sigma(\eta_{\text{th}})$. The influence of $\sigma(\eta_{\text{HPT}})$ on $\sigma(\eta_{\text{o}})$ is more dominant but a similar effect can be observed at larger $\sigma(T_{t,40})$.

Figure E.1: Iso-lines of an interpolation of a grid of 30 Monte Carlo simulations illustrating the relative effect of uncertainties in HPT efficiency $\eta_{\text{HPT}}$ and inlet temperature $T_{t,40}$ on engine performance.
# List of Figures

1.1 Turbofan components and cycle process ........................................... 2
1.2 Thermal load and inlet conditions of the high pressure turbine ............ 4
1.3 Coupled and uncoupled approach to CTI ......................................... 6
1.4 Variability of turbine inlet traverse .............................................. 7
1.5 Uncertainty in LSTR swirl field .................................................. 8
1.6 Robust design of high pressure turbines ......................................... 9

2.1 RQL combustion chamber .......................................................... 13
2.2 LSTR combustor swirl and bubble vortex breakdown ......................... 15
2.3 Tangential swirl migration between swirler and turbine inlet ............... 16
2.4 Turbine stage thermodynamics .................................................. 19
2.5 Turbine vane secondary flows ................................................. 22
2.6 Determining HTC from two measurements ................................... 25
2.7 Preferential Heating ............................................................. 28
2.8 Swirl induced vortex structures at stator LE .................................. 31

3.1 Turbulent energy cascade .......................................................... 40
3.2 High pressure turbine CFD boundary conditions .............................. 50
3.3 Discontinuity at CTI interface .................................................. 52
3.4 LSTR: Numerical set-up of NGV1 row ....................................... 54
3.5 LSTR: ME01 and ME02, influence of mesh resolution ...................... 57
3.6 LSTR: NGV1 blade loading at axial inflow .................................. 58
3.7 LSTR: Distribution of $\eta_{FC}$ and Nu at the hub end wall ................ 59
3.8 LSTR: Comparison of film cooling effectiveness for axial inflow ......... 59
3.9 LSTR: Comparison of film cooling effectiveness for axial inflow ......... 60
3.10 LSTR: Local blowing ratios across the RIDN holes ......................... 61
3.11 LSTR: Comparison of Nusselt number for axial inflow .................... 61
3.12 LSTR: ME01 and ME02, influence of turbulence model .................. 62
3.13 LSTR: NGV1 blade loading at swirling inflow ................................ 63
3.14 LSTR: Local blowing rate for swirling inflow ................................ 64
3.15 LSTR: Comparison of film cooling effectiveness for swirling inflow .... 65
3.16 LSTR: Comparison of Nusselt number for swirling inflow ................ 66
3.17 Statistical properties ............................................................ 70
3.18 Workflow of Monte Carlo analysis with point collocation ................. 72
3.19 Monte Carlo simulation of engine cycle parameters ....................... 73
3.20 Workflow of gPC analysis with point collocation ................. 75

4.1 Lean burn traverses ........................................... 78
4.2 Structure of the interface parameterisation model .................. 79
4.3 Swirler vortex modelling ....................................... 81
4.4 Periodicity of boundaries ....................................... 82
4.5 Wall tangential flow ........................................... 83
4.6 Modelling of flow entrainment .................................. 85
4.7 Parameterisation of the temperature profile ......................... 87
4.8 LSTR: Matched inlet traverse ................................... 92
4.9 RRD-HPT: Clocking position and matching of inlet conditions .. 93
4.10 RRD-HPT: Comparison of stage performance over clocking ..... 94
4.11 RRD-HPT: Nusselt number for datum and matched inlet conditions 95
4.12 RRD-HPT: Wall temperature for datum and matched inlet conditions 96
4.13 Uncertainty quantification workflow ............................ 97

5.1 RRD-HPT: Generic swirl inlet condition .......................... 102
5.2 RRD-HPT: Efficiency deficit at increasing swirl number ........ 104
5.3 RRD-HPT: Efficiency and losses at different swirl clocking positions. 105
5.4 RRD-HPT: Rotor incidence and entropy loss coefficient .......... 106
5.5 RRD-HPT: Lift deficit at increasing swirl number ................ 107
5.6 RRD-HPT: Swirl induced radial pressure gradients ............... 108
5.7 RRD-HPT: Total pressure profiles at NGV1 exit .................. 110
5.8 RRD-HPT: Interaction of inlet swirl and flow to stagnation line 111
5.9 RRD-HPT: Correlation of inlet swirl with LE vortex radial position . 113
5.10 LSTR: NGV flow field for axial and swirling inflow ............ 114
5.11 LSTR: Swirl induced vortex at LE ............................. 115
5.12 LSTR: Uncertainty propagation to hub film cooling and heat transfer 118
5.13 LSTR: Uncertainty propagation to vane film cooling and heat transfer 119
5.14 LSTR: Influence of inlet total pressure field ..................... 120
5.15 E3E: Geometry, numerical set-up and mesh ..................... 121
5.16 E3E: Mesh study ................................................ 123
5.17 E3E: Secondary flows and wall temperature of baseline configuration 125
5.18 E3E: Comparison of datum and match inlet conditions .......... 126
5.19 E3E: Wall temperature of matched inlet traverse ................ 127
5.20 E3E: Temperature scaling ...................................... 129
5.21 E3E: Inlet temperature correction .............................. 130
5.22 E3E: 0D correlation matrices .................................. 132
5.23 E3E: Influence of radial hot spot position on rotor loading and losses 133
5.24 E3E: Uncertainty propagation to vane temperature and heat transfer 135
5.25 E3E: Uncertainty propagation to hub temperature and heat transfer . 136
5.26 E3E: Interaction of inlet swirl with hot streak migration . . . . . 138

A.1 Mechanisms affecting efficiency gains . . . . . . . . . . . . . . . . . . . 145

B.1 Scaling the CTI interface conditions . . . . . . . . . . . . . . . . . . . . . 151
B.2 Temperature traverse of case II . . . . . . . . . . . . . . . . . . . . . . . 153
B.3 Difference in enthalpy inflow for equal mean total temperatures . . 155
B.4 Hammersley sampling . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 156
B.5 Latin-Hypercube sampling . . . . . . . . . . . . . . . . . . . . . . . . . . . 157
B.6 Comparison of gPC settings . . . . . . . . . . . . . . . . . . . . . . . . . . 159

C.1 Lamb-Oseen vortex . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 161
C.2 Definition of the pseudo swirl number . . . . . . . . . . . . . . . . . . . 162
C.3 Circumferential distribution of entrained flow . . . . . . . . . . . . . . 163
C.4 Radial distribution of radially entrained flow . . . . . . . . . . . . . . 164
C.5 Radial distribution of circumferentially entrained flow . . . . . . . . 165
C.6 Functions for flow entrainment modelling . . . . . . . . . . . . . . . . . . 166
C.7 Parameter linking . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 170
C.8 RRD-HPT: Numerical set-up and mesh . . . . . . . . . . . . . . . . . . . . 172
C.9 RRD-HPT: Mesh study . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 173
C.10 RRD-HPT: Matching of inlet traverses and NGV exit profiles . . . . 174
C.11 RRD-HPT: Matching of test case 1 . . . . . . . . . . . . . . . . . . . . . 176
C.12 RRD-HPT: Matching of test case 2 . . . . . . . . . . . . . . . . . . . . . 177
C.13 RRD-HPT: Matching of test case 3 . . . . . . . . . . . . . . . . . . . . . 179
C.14 RRD-HPT: Matching of test case 4 . . . . . . . . . . . . . . . . . . . . . 179
C.15 RRD-HPT: Matching of test case 5 . . . . . . . . . . . . . . . . . . . . . 180
C.16 RRD-HPT: Nusselt number for datum and matched inlet conditions . 181
C.17 RRD-HPT: Wall temperature for datum and matched inlet conditions 181
C.18 RRD-HPT: Divergence of inlet traverse correction . . . . . . . . . . . . 182

D.1 RRD-HPT: Efficiency deficit at increasing swirl number . . . . . . . . 185
D.2 RRD-HPT: Total pressure profiles at NGV1 exit . . . . . . . . . . . . . 186
D.3 RRD-HPT: Influence of turbulence model on pressure loss . . . . . . 187
D.4 RRD-HPT: Approximation of efficiency loss from inlet swirl . . . . . . 188
D.5 LSTR (Swirl Generator): Numerical set-up of swirl generator . . . . . 189
D.6 LSTR: Uncertainty of vane film cooling with fixed inlet pressure profile190
D.7 LSTR: Uncertainty propagation to vane heat transfer . . . . . . . . . . . 191
D.8 E3E: Effect of turbulence model on swirl-induced structure in E3E . 192
D.9 E3E: Influence of turbulence modelling on wall temperature distribution192
D.10 E3E: Nusselt number for the datum and matched 2D inlet conditions 193
D.11 Derivation of uncertainties of temperature field parameters . . . . . . 194
D.12 E3E: Correlation matrices (gPC without temperature correction) . . 196
D.13 E3E: Correlation matrices (MCS with temperature correction) . . . 196
D.14 E3E: Temperature weighted standard deviation of wall temperature . 197
D.15 E3E: Correlations of inlet traverse parameters with wall temperature 198
D.16 E3E: Correlations of inlet traverse parameters with Nusselt number . 199

E.1 Impact of uncertainties in HPT efficiency and inlet temperature . . . 202
List of Tables

3.1 LSTR: Numerical Set-up ............................................. 56
4.1 List of Parameters for Combustor Exit Flow Model ............. 90
5.1 RRD-HPT: Parameters and Distributions of Monte Carlo Runs ...... 103
5.2 LSTR: Uncertainty Propagation through NGV1 ....................... 117
5.3 E3E: Numerical Set-up ............................................ 122
5.4 E3E: Uncertainty Propagation through the HPT ................. 131
B.1 Characteristic Cycle Properties of Turbofan Engine ............ 158
C.1 Linking of First and Second Order Parameters .................. 169
C.2 Parameters open in the Matching of Combustor Exit Traverses ........ 171
C.3 RRD-HPT: Numerical Set-up ..................................... 172
C.4 Matched Swirl Parameters of the Combustor Exit Flow Model .......... 175
D.1 RRD-HPT: Effect of TBR on efficiency predictions ............... 187
D.2 LSTR (Swirl Generator): Numerical Set-up ......................... 189
D.3 E3E: Uncertainty Propagation through the HPT ................. 195
E.1 Datum Values of Engine Cycle Parameters in Monte Carlo Analysis . 201
## List of Symbols

### Lower Case Latin Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$a$</td>
<td>m/s</td>
<td>Speed of sound</td>
</tr>
<tr>
<td>$a$, $b$, $c$</td>
<td></td>
<td>Generic numerical values or constants</td>
</tr>
<tr>
<td>$b$</td>
<td></td>
<td>Source vector</td>
</tr>
<tr>
<td>$b_{ij}$</td>
<td></td>
<td>Anisotropic tensor</td>
</tr>
<tr>
<td>$c_p$</td>
<td>J/kg K</td>
<td>Specific heat capacity at constant pressure</td>
</tr>
<tr>
<td>$c_h$</td>
<td>m</td>
<td>Chord length</td>
</tr>
<tr>
<td>$d$</td>
<td></td>
<td>Swirl rotation direction</td>
</tr>
<tr>
<td>$f$</td>
<td></td>
<td>Fuel-air-ratio</td>
</tr>
<tr>
<td>$f_{\mu}$</td>
<td></td>
<td>Damping function</td>
</tr>
<tr>
<td>$g$</td>
<td>m/s$^2$</td>
<td>Gravitation</td>
</tr>
<tr>
<td>$h$</td>
<td>W/m$^2$ K</td>
<td>Heat transfer coefficient</td>
</tr>
<tr>
<td>$h_{rel}$</td>
<td></td>
<td>Relative channel height</td>
</tr>
<tr>
<td>$k$</td>
<td>m$^2$/s$^2$</td>
<td>Turbulent kinetic energy</td>
</tr>
<tr>
<td>$k_{\text{rel}}$</td>
<td>W/m K</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>$m$</td>
<td>kg/s</td>
<td>Mass flow</td>
</tr>
<tr>
<td>$\dot{m}_{\text{bypass}}$</td>
<td>kg/s</td>
<td>Engine bypass stream mass flow</td>
</tr>
<tr>
<td>$\dot{m}_{\text{core}}$</td>
<td>kg/s</td>
<td>Engine core mass flow</td>
</tr>
<tr>
<td>$\dot{m}_f$</td>
<td>kg/s</td>
<td>Fuel mass flow</td>
</tr>
<tr>
<td>$\dot{m}_{\text{ox}}$</td>
<td>kg/s</td>
<td>Oxidiser mass flow</td>
</tr>
<tr>
<td>$n$</td>
<td></td>
<td>Dimension of inputs to probabilistic system</td>
</tr>
<tr>
<td>$n_{\text{p}}$</td>
<td></td>
<td>gPC oversampling ratio</td>
</tr>
<tr>
<td>$p$</td>
<td></td>
<td>Polynomial order</td>
</tr>
<tr>
<td>$\dot{p}$</td>
<td>Pa</td>
<td>Pressure</td>
</tr>
<tr>
<td>$q$</td>
<td>W/m$^2$</td>
<td>Specific heat flux</td>
</tr>
<tr>
<td>$r$</td>
<td>m</td>
<td>Radial coordinate (with respect to machine axis)</td>
</tr>
<tr>
<td>$r_{\text{rel}}$</td>
<td></td>
<td>Recovery factor</td>
</tr>
</tbody>
</table>
Lower Case Latin Symbols

\begin{itemize}
  \item \(s\) \(\text{m}\) Arc length
  \item \(t\) \(\text{s}\) Time
  \item \(u\) \(\text{m/s}\) Velocity
  \item \(u^+\) \(\text{m/s}\) Wall shear velocity
  \item \(w\) \(\text{m/s}\) Velocity (relative frame of reference)
  \item \(x\) Solution vector
  \item \(x, y, z\) \(\text{m}\) Spatial coordinate
  \item \(x, y\) Deterministic inputs and outputs to a system
  \item \(x_k, y_k\) Weights of polynomial expansions
  \item \(y^+\) Non-dimensional wall distance
\end{itemize}

Upper Case Latin Symbols

\begin{table}[h]
\begin{tabular}{lll}
Symbol & Unit & Description \\
\hline
\(A\) & \(\text{m}^2\) & Area \\
\(\text{—“—}\) & - & Event \\
\(A\) & Coefficient matrix \\
\(C\) & Constant \\
\(\text{—“—}\) & \(\text{m/s}\) & Velocity (absolute frame of reference) \\
\(C\) & Carbon \\
\(C_k\) & - & Weighting factor \\
\(C_u\) & \(\text{m/s}\) & Circumferential velocity (absolute frame of reference) \\
\(D\) & \(\text{m}\) & Diameter \\
\(\Theta_{ij}\) & \(\text{m}^2/\text{s}^3\) & Turbulent diffusion tensor \\
\(E\) & \(\text{J}\) & Energy \\
\(E_{\text{kin}}\) & \(\text{J}\) & Kinetic energy \\
\(F_n\) & \(\text{N}\) & Engine net thrust \\
\(F_i\) & - & Blending function \\
\(H\) & \(\text{m}\) & Channel height \\
\(\text{—“—}\) & \(\text{J}\) & Enthalpy \\
\(H\) & Hydrogen \\
\(\Delta H\) & \(\text{J}\) & Creep activation energy \\
\(I\) & - & Momentum ratio \\
\(I, \text{II, III}\) & Tensor invariants \\
\(L\) & \(\text{m}\) & Length \\
\(\text{—“—}\) & \(\text{N}\) & Lift \\
\(\mathcal{L}\) & \(\text{m}\) & Turbulent length scale \\
\end{tabular}
\end{table}
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Ma</td>
<td>Mach number</td>
</tr>
<tr>
<td>Ma_{is}</td>
<td>Isentropic Mach number</td>
</tr>
<tr>
<td>MFR</td>
<td>Mass flow ratio</td>
</tr>
<tr>
<td>N</td>
<td>Number of samples</td>
</tr>
<tr>
<td>N_{pop}</td>
<td>Population of a generation of the optimisation algorithm</td>
</tr>
<tr>
<td>N_{sec}</td>
<td>Number of flow sectors</td>
</tr>
<tr>
<td>N_{sw}</td>
<td>Number of superimposed swirl cores</td>
</tr>
<tr>
<td>N</td>
<td>Nitrogen</td>
</tr>
<tr>
<td>N_{r},N_{t}</td>
<td>Grid nodes in radial and circumferential direction</td>
</tr>
<tr>
<td>Nu</td>
<td>Nusselt number</td>
</tr>
<tr>
<td>O</td>
<td>Oxygen</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>P</td>
<td>Power</td>
</tr>
<tr>
<td>P_{th}</td>
<td>Thermal power</td>
</tr>
<tr>
<td>\mathbf{P}_{ij}</td>
<td>Production tensor of turbulent energy</td>
</tr>
<tr>
<td>Pr_t</td>
<td>Turbulent Prandtl number</td>
</tr>
<tr>
<td>Q_R</td>
<td>Fuel heating value</td>
</tr>
<tr>
<td>R</td>
<td>Radial coordinate (with respect to swirl centre)</td>
</tr>
<tr>
<td>R_{curv}</td>
<td>Local radius of curvature</td>
</tr>
<tr>
<td>R_{cr}</td>
<td>Creep process rate</td>
</tr>
<tr>
<td>\mathbf{R}_{ij}</td>
<td>Rotation tensor</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>Re_y</td>
<td>Reynolds number based on wall normal coordinate</td>
</tr>
<tr>
<td>S</td>
<td>Pseudo-swirl number</td>
</tr>
<tr>
<td>S_s</td>
<td>Surface</td>
</tr>
<tr>
<td>S_0</td>
<td>Swirl number</td>
</tr>
<tr>
<td>S</td>
<td>Sulphur</td>
</tr>
<tr>
<td>S_\Phi</td>
<td>Source term</td>
</tr>
<tr>
<td>T</td>
<td>Temperature</td>
</tr>
<tr>
<td>T^*</td>
<td>Threshold temperature</td>
</tr>
<tr>
<td>\mathcal{T}</td>
<td>Turbulent time scale</td>
</tr>
<tr>
<td>Tu</td>
<td>Turbulence Intensity</td>
</tr>
<tr>
<td>U</td>
<td>Rotor wheel speed</td>
</tr>
<tr>
<td>U_{ij}</td>
<td>Strain rate tensor</td>
</tr>
<tr>
<td>V</td>
<td>Volume</td>
</tr>
<tr>
<td>V_0</td>
<td>Aircraft flight velocity</td>
</tr>
<tr>
<td>Symbol</td>
<td>Unit</td>
</tr>
<tr>
<td>--------</td>
<td>------------</td>
</tr>
<tr>
<td>$W$</td>
<td>m/s</td>
</tr>
<tr>
<td>$X, Y$</td>
<td></td>
</tr>
<tr>
<td>$\dot{X}, \dot{Y}$</td>
<td></td>
</tr>
</tbody>
</table>

**Lower Case Greek Symbols**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\alpha$</td>
<td></td>
<td>Bypass ratio</td>
</tr>
<tr>
<td>$\beta$</td>
<td>°</td>
<td>Flow angle (absolute frame of reference)</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>-</td>
<td>Ratio of specific heats</td>
</tr>
<tr>
<td>$\delta_{ij}$</td>
<td></td>
<td>Kronecker delta</td>
</tr>
<tr>
<td>$\varepsilon$</td>
<td>m$^2$/s$^3$</td>
<td>Turbulent dissipation rate</td>
</tr>
<tr>
<td>$\varepsilon_{ij}$</td>
<td>m$^2$/s$^3$</td>
<td>Turbulent dissipation tensor</td>
</tr>
<tr>
<td>$\zeta$</td>
<td>-</td>
<td>Parameter for radial positioning</td>
</tr>
<tr>
<td>$\eta$</td>
<td>%</td>
<td>Generic system</td>
</tr>
<tr>
<td>$\eta'$</td>
<td>%</td>
<td>Isentropic efficiency</td>
</tr>
<tr>
<td>$\eta_{FC}$</td>
<td>%</td>
<td>Polytropic efficiency</td>
</tr>
<tr>
<td>$\eta_o$</td>
<td>%</td>
<td>Film cooling effectiveness</td>
</tr>
<tr>
<td>$\eta_{th}$</td>
<td>%</td>
<td>Engine overall efficiency</td>
</tr>
<tr>
<td>$\eta_{pr}$</td>
<td>%</td>
<td>Engine thermal efficiency</td>
</tr>
<tr>
<td>$\eta_{le}$</td>
<td>%</td>
<td>Engine propulsive efficiency</td>
</tr>
<tr>
<td>$\theta$</td>
<td>°</td>
<td>Circumferential coordinate</td>
</tr>
<tr>
<td>$\theta_{pitch}$</td>
<td>°</td>
<td>Clocking position</td>
</tr>
<tr>
<td>$\theta$</td>
<td>°</td>
<td>Non-dimensional temperature</td>
</tr>
<tr>
<td>$\mu$</td>
<td>kg/m s</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>m/s$^2$</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>$\nu_t$</td>
<td>m/s$^2$</td>
<td>Turbulent viscosity</td>
</tr>
<tr>
<td>$\xi$</td>
<td>-</td>
<td>Averaging weight</td>
</tr>
<tr>
<td>$\pi$</td>
<td>-</td>
<td>Basis for polynomial expansion</td>
</tr>
<tr>
<td>$\rho$</td>
<td></td>
<td>Mathematical constant ($\pi = 3.14159 \ldots$)</td>
</tr>
</tbody>
</table>

212 List of Symbols
—"— kg/m³ Density

σ Standard deviation of Normal distribution
σₜ - Turbulent Prandtl number
σₓ - Hot spot scaling parameter
σᵧ - Hot spot scaling parameter
τₔ N/m² Wall shear stress
τ - Temperature ratio
ϕ - Equivalence ratio
—"— ° Hot streak rotation angle
χ - Swirl stretching parameter
ψ - Shape parameter
ω - Element of sample space
—"— rpm Rotational speed of rotor
—"— 1/s Specific turbulent dissipation rate
—"— 1/s Vorticity

Upper Case Greek Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Γ</td>
<td>m²/s</td>
<td>Circulation</td>
</tr>
<tr>
<td>—&quot;—</td>
<td>kg K⁰.⁵/Pa s</td>
<td>Turbine capacity</td>
</tr>
<tr>
<td>Γₚ</td>
<td>m</td>
<td>Generic diffusion coefficient</td>
</tr>
<tr>
<td>Δ</td>
<td>m</td>
<td>Grid spacing</td>
</tr>
<tr>
<td>Πᵯᵢⱼ</td>
<td>m²/s³</td>
<td>Pressure strain correlation</td>
</tr>
<tr>
<td>Πᵯᵣᵢⱼ</td>
<td>m²/s³</td>
<td>Rapid term</td>
</tr>
<tr>
<td>Πᵯₑᵢⱼ</td>
<td>m²/s³</td>
<td>Return term</td>
</tr>
<tr>
<td>Φ</td>
<td>-</td>
<td>Generic physical quantity</td>
</tr>
<tr>
<td>Ψ</td>
<td>-</td>
<td>Polynomial</td>
</tr>
<tr>
<td>Ω</td>
<td>-</td>
<td>Sample space</td>
</tr>
<tr>
<td>Ωᵢⱼ</td>
<td>1/s</td>
<td>Rotation tensor</td>
</tr>
</tbody>
</table>

Subscripts

<table>
<thead>
<tr>
<th>Subscript</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>□∞</td>
<td>Free stream/bulk conditions</td>
</tr>
<tr>
<td>□₀</td>
<td>Before turbine stator</td>
</tr>
<tr>
<td>—&quot;—</td>
<td>Initial (uncorrected) field</td>
</tr>
<tr>
<td>Symbol</td>
<td>Description</td>
</tr>
<tr>
<td>--------</td>
<td>-------------</td>
</tr>
<tr>
<td>$\square_1$</td>
<td>Before turbine stator</td>
</tr>
<tr>
<td>$\square_{10}$</td>
<td>First order parameter</td>
</tr>
<tr>
<td>$\square_2$</td>
<td>Behind turbine rotor</td>
</tr>
<tr>
<td>$\square_{20}$</td>
<td>Second order parameter</td>
</tr>
<tr>
<td>$\square_3$</td>
<td>Behind the compressor</td>
</tr>
<tr>
<td>$\square_{40}$</td>
<td>Behind the combustor</td>
</tr>
<tr>
<td>$\square_{45}$</td>
<td>Behind the high pressure turbine</td>
</tr>
<tr>
<td>$\square_5$</td>
<td>Behind the turbine</td>
</tr>
<tr>
<td>$\square_{ax}$</td>
<td>In axial direction</td>
</tr>
<tr>
<td>$\square_{ad}$</td>
<td>Adiabatic</td>
</tr>
<tr>
<td>$\square_{BC}$</td>
<td>Boundary condition</td>
</tr>
<tr>
<td>$\square_{bg}$</td>
<td>Background</td>
</tr>
<tr>
<td>$\square_c$</td>
<td>Coolant</td>
</tr>
<tr>
<td>$\square_{CDS}$</td>
<td>CDS coefficient</td>
</tr>
<tr>
<td>$\square_{DP}$</td>
<td>Design point</td>
</tr>
<tr>
<td>$\square_{eff}$</td>
<td>Effective</td>
</tr>
<tr>
<td>$\square_{gh}$</td>
<td>Ghost vortex</td>
</tr>
<tr>
<td>$\square_h$</td>
<td>At the hub</td>
</tr>
<tr>
<td>$\square_{HPT}$</td>
<td>High pressure turbine</td>
</tr>
<tr>
<td>$\square_{hs}$</td>
<td>Hot spot</td>
</tr>
<tr>
<td>$\square_i$</td>
<td>Isentropic</td>
</tr>
<tr>
<td>$\square_i$</td>
<td>$i^{th}$ hot spot</td>
</tr>
<tr>
<td>$\square_{ij}$</td>
<td>Spatial coordinate $i</td>
</tr>
<tr>
<td>$\square_{in}$</td>
<td>At turbine inlet</td>
</tr>
<tr>
<td>$\square_{LE}$</td>
<td>At the vane leading edge</td>
</tr>
<tr>
<td>$\square_m$</td>
<td>Mirrored vortex</td>
</tr>
<tr>
<td>$\square_{max}$</td>
<td>Maximal value</td>
</tr>
<tr>
<td>$\square_{out}$</td>
<td>At stage or row outlet</td>
</tr>
<tr>
<td>$\square_{per}$</td>
<td>Periodic sector</td>
</tr>
<tr>
<td>$\square_{rad}$</td>
<td>In radial direction</td>
</tr>
<tr>
<td>$\square_{ref}$</td>
<td>Reference</td>
</tr>
<tr>
<td>$\square_s$</td>
<td>At the shroud</td>
</tr>
<tr>
<td>$\square_{sec}$</td>
<td>Sector</td>
</tr>
<tr>
<td>$\square_{stoic}$</td>
<td>Stoichiometric</td>
</tr>
<tr>
<td>$\square_{str}$</td>
<td>Stretched vortex</td>
</tr>
<tr>
<td>$\square_{sw}$</td>
<td>Swirl</td>
</tr>
<tr>
<td>$\square_{SST}$</td>
<td>Constant of the SST model</td>
</tr>
<tr>
<td>$\square_t$</td>
<td>Total quantity</td>
</tr>
<tr>
<td>$\square_{tan}$</td>
<td>In tangential direction</td>
</tr>
<tr>
<td>$\square_{tw}$</td>
<td>Tangential to end wall</td>
</tr>
</tbody>
</table>
UDS coefficient
At the wall
In wall normal direction
With respect to the Nozzle Guide Vane
Constant of the $k-\omega$ model
Constant of the $k-\epsilon$ model

Superscripts

<table>
<thead>
<tr>
<th>Superscript</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>${bg}$</td>
<td>Temperature background profile</td>
</tr>
<tr>
<td>${ec}$</td>
<td>Endwall convergence</td>
</tr>
<tr>
<td>${fe}$</td>
<td>Flow entrainment</td>
</tr>
<tr>
<td>${hs}$</td>
<td>Hot spot</td>
</tr>
<tr>
<td>${ref}$</td>
<td>Reference field</td>
</tr>
<tr>
<td>${sw}$</td>
<td>Swirl flow</td>
</tr>
</tbody>
</table>

Operators

<table>
<thead>
<tr>
<th>Operator</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\propto$</td>
<td>Proportional to</td>
</tr>
<tr>
<td>$\approx$</td>
<td>Approximately equal</td>
</tr>
<tr>
<td>$:=!$</td>
<td>Equal by definition</td>
</tr>
<tr>
<td>$\overset{!}{=}$</td>
<td>Equality enforced by scaling</td>
</tr>
<tr>
<td>$f(x)$</td>
<td>Probability density Function</td>
</tr>
<tr>
<td>$F(x)$</td>
<td>Cumulative density function</td>
</tr>
<tr>
<td>$f(\cdot), g(\cdot), h(\cdot)$</td>
<td>Generic functions</td>
</tr>
<tr>
<td>$\langle f, g \rangle$</td>
<td>Orthogonal functions $f$ and $g$</td>
</tr>
<tr>
<td>exp$(x)$</td>
<td>Exponential function</td>
</tr>
<tr>
<td>$f_W(x)$</td>
<td>Lambert $W$ function</td>
</tr>
<tr>
<td>$\hat{f}(\cdot)$</td>
<td>Normalised function $f$</td>
</tr>
<tr>
<td>$\mathbf{x}$</td>
<td>Vector</td>
</tr>
<tr>
<td>$\mathbf{X}$</td>
<td>Matrix</td>
</tr>
<tr>
<td>$\bar{\Phi}$</td>
<td>Normalised quantity $\Phi$</td>
</tr>
<tr>
<td>$\Phi'$</td>
<td>Turbulent fluctuation of $\Phi$</td>
</tr>
<tr>
<td>$\Phi^*$</td>
<td>Threshold/corrected value of $\Phi$</td>
</tr>
</tbody>
</table>
\( \rho^* \) Significant correlation
\( \dot{\Phi} \) Flux of \( \Phi \)
\( \overline{\Phi} \) Average of \( \Phi \)
\( |\Phi| \) Absolute value of \( \Phi \)
\( \max(\Phi) \) Maximum of \( \Phi \)
\( \lfloor \Phi \rfloor \) \( \Phi \) rounded down to the next integer
\( \delta(\Phi) \) Incremental difference of \( \Phi \)
\( \Delta(\Phi) \) Finite difference of \( \Phi \)
\( \theta(x) \) At the order of \( x \)
\( E[X] \) Expectation of \( X \)
\( \text{Cov}(X, Y) \) Covariance of \( X \) and \( Y \)
\( \text{Var}(X) \) Variance of \( X \)
\( \sum_{i=x}^{N} \) Sum from \( i = x \) to \( i = N \)
\( \lim_{x \to \infty} \) In the limit of infinite \( x \)
\( \frac{\partial \Phi}{\partial x} \) Partial derivative of \( \Phi \) by \( x \)
\( \nabla \) Nabla operator
\( \int_{x_0}^{x_1} f(x) \, dx \) Integral of the function \( f(x) \) between \( x_0 \) and \( x_1 \)

**Abbreviations**

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0D</td>
<td>Zero-Dimensional</td>
</tr>
<tr>
<td>1D</td>
<td>One-Dimensional</td>
</tr>
<tr>
<td>2D</td>
<td>Two-Dimensional</td>
</tr>
<tr>
<td>3D</td>
<td>Three-Dimensional</td>
</tr>
<tr>
<td>avg.</td>
<td>Average</td>
</tr>
<tr>
<td>Ax</td>
<td>Axial inflow condition</td>
</tr>
<tr>
<td>BC</td>
<td>Boundary Condition</td>
</tr>
<tr>
<td>BR</td>
<td>Blowing Ratio</td>
</tr>
<tr>
<td>CDF</td>
<td>Cumulative Density Function</td>
</tr>
<tr>
<td>CDS</td>
<td>Central Differencing Scheme</td>
</tr>
<tr>
<td>cf.</td>
<td>Confer (compare)</td>
</tr>
<tr>
<td>CFD</td>
<td>Computational Fluid Dynamics</td>
</tr>
<tr>
<td>CFL</td>
<td>Courant, Friedrichs &amp; Lewy</td>
</tr>
<tr>
<td>CHT</td>
<td>Conjugate Heat Transfer</td>
</tr>
<tr>
<td>circ.</td>
<td>Circumferential</td>
</tr>
<tr>
<td>const.</td>
<td>Constant</td>
</tr>
<tr>
<td>CTI</td>
<td>Combustor Turbine Interaction</td>
</tr>
<tr>
<td>Abbreviation</td>
<td>Full Form</td>
</tr>
<tr>
<td>--------------</td>
<td>-----------</td>
</tr>
<tr>
<td>DNS</td>
<td>Direct Numerical Simulation</td>
</tr>
<tr>
<td>DR</td>
<td>Density Ratio</td>
</tr>
<tr>
<td>e.g.</td>
<td><em>Exempli gratia</em> (for example)</td>
</tr>
<tr>
<td>EARSM</td>
<td>Explicit Algebraic Reynolds Stress Model</td>
</tr>
<tr>
<td>EVM</td>
<td>Eddy Viscosity Model</td>
</tr>
<tr>
<td>EVR</td>
<td>Equi-Volume Ratio</td>
</tr>
<tr>
<td>E3E</td>
<td>Engine Environmental friendliness, Efficiency, Economy</td>
</tr>
<tr>
<td>EPFL</td>
<td>École Polytechnique Fédérale de Lausanne</td>
</tr>
<tr>
<td>exp.</td>
<td>Experimental</td>
</tr>
<tr>
<td>FACTOR</td>
<td>Full Aerothermal Combustor-Turbine interctions Research</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>fluct.</td>
<td>Fluctuation</td>
</tr>
<tr>
<td>FVM</td>
<td>Finite Volume Method</td>
</tr>
<tr>
<td>geo.</td>
<td>Interface Geometry</td>
</tr>
<tr>
<td>HPT</td>
<td>High Pressure Turbine</td>
</tr>
<tr>
<td>HTC</td>
<td>Heat Transfer Coefficient</td>
</tr>
<tr>
<td>i.e.</td>
<td><em>Id est</em> (that is)</td>
</tr>
<tr>
<td>IPCC</td>
<td>Intergovernmental Panel on Climate Change</td>
</tr>
<tr>
<td>ICAO</td>
<td>International Civil Aviation Organisation</td>
</tr>
<tr>
<td>LSTR</td>
<td>Large Scale Turbine Rig</td>
</tr>
<tr>
<td>LE</td>
<td>Leading Edge</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>LHS</td>
<td>Latin Hypercube Sampling</td>
</tr>
<tr>
<td>LPP</td>
<td>Lean, Premixed, Prevaporised</td>
</tr>
<tr>
<td>LRR</td>
<td>Launder, Reece &amp; Rodi</td>
</tr>
<tr>
<td>Max.</td>
<td>Maximum</td>
</tr>
<tr>
<td>MCS</td>
<td>Monte Carlo Sampling</td>
</tr>
<tr>
<td>Min.</td>
<td>Minimum</td>
</tr>
<tr>
<td>Mio.</td>
<td>Million</td>
</tr>
<tr>
<td>ME</td>
<td>Measurement Plane (<em>Messebene</em>)</td>
</tr>
<tr>
<td>MFR</td>
<td>Mass Flow Ratio</td>
</tr>
<tr>
<td>MR</td>
<td>Mach number Ratio</td>
</tr>
<tr>
<td>neg.</td>
<td>Negative</td>
</tr>
<tr>
<td>NGV</td>
<td>Nozzle Guide Vane</td>
</tr>
<tr>
<td>OC</td>
<td>Operating Conditions</td>
</tr>
<tr>
<td>(L)OTDF</td>
<td>(Local) Overall Temperature Distortion Factor</td>
</tr>
<tr>
<td>par.</td>
<td>Parameters</td>
</tr>
<tr>
<td>(g)PC</td>
<td>(Generalised) Polynomial Chaos</td>
</tr>
</tbody>
</table>
PDF  Probability Density Function
pos.  Positive
PS  Pressure Side
RANS  Reynolds-Averaged Navier-Stokes
RIDN  Rear Inner Discharge Nozzle
RODN  Rear Outer Discharge Nozzle
rpm  Revolutions Per Minute
RQL  Rich-burn Quick-quench Lean-burn
RRD  Rolls-Royce Deutschland Ltd & Co KG
RRD-HPT  Rolls-Royce Deutschland High Pressure Turbine
RSM  Reynolds Stress Model
RSM-BSL  CFX Baseline Reynolds Stress Model
(L)RTDF  (Local) Radial Temperature Distortion Factor
SAS  Scale Adaptive Simulation
SS  Suction Side
SSG  Speziale, Sakar & Gatski
SST  Shear Stress Transport
SwL  Swirl clocked to Leading edge
SwP  Swirl clocked to Passage
TATEF  Turbine Aero-Thermal External Flows
Temp.  Temperature
TU  Technical University
TBC  Thermal Barrier Coating
TBR  Transient Blade Row interaction
UDS  Upwind Differencing Scheme
UHC  Unburned Hydrocarbons
UQ  Uncertainty Quantification
v.  Version
vs.  Versus
Bibliography


Bibliography 227


