Numerical Investigation of Radial Turbines Subject to Pulsating Flow

ROBERTO MOSCA
Numerical Investigation of Radial Turbines Subject to Pulsating Flow

ROBERTO MOSCA

Academic Dissertation which, with due permission of the KTH Royal Institute of Technology, is submitted for public defence for the Degree of Doctor of Philosophy on Friday the 7th October 2022, at 10:00 a.m. in F3, Lindstedsvägen 26, Stockholm

Doctoral Thesis in Engineering Mechanics
KTH Royal Institute of Technology
Stockholm, Sweden 2022
Numerical Investigation of Radial Turbines Subject to Pulsating Flow

Roberto Mosca
CCGEx, KTH Royal Institute of Technology, Department of Engineering Mechanics SE-100 44 Stockholm, Sweden

Abstract

In the optic of a more sustainable society, research and development of highly efficient fluid machines represent a fundamental process to satisfy the rapidly growing energy needs of the modern world. Radial turbines are characterized by higher efficiencies for a larger range of inflow conditions compared to axial turbines. Due to this favorable characteristic, they find their natural application in turbocharger systems, where the flow is inherently unsteady due to the engine reciprocating. In a turbocharged engine, to exploit the residual energy contained in the exhaust gases, the radial turbine is fed by the exhaust gases from the cylinders of the engine. The particular inflow conditions to which a turbocharger turbine is exposed, i.e. pulsating flow and high gas temperatures, make the turbocharger turbine a unique example in the turbomachinery field. Indeed, pulsating flow causes performance deviations from quasi-steady to pulsating flow conditions, while heat transfer deteriorates the turbine performance. Modeling correctly these phenomena is essential to enhance turbocharger-engine matching. The problem is further complicated since, due to the geometrical diversity of the different parts of the system, each component represents a stand-alone problem both in terms of flow characteristics and design optimization.

In this thesis, high-fidelity numerical simulations are used to characterize the performance of a single-entry radial turbine applied in a commercial 4-cylinder engine for a passenger car under engine-like conditions. By treating the hot-side system as a stand-alone device, parametrization of the pulse shape imposed as inlet boundary conditions has let to highlight specific trends of the system response to pulse amplitude and frequency variations. Reduced-order models to predict the deviations of the turbine performance from quasi-steady to pulsating flow conditions are developed. At first, a simple algebraic model demonstrates the proportionality between the intensity of the deviations and the normalized reduced frequency. Then, a neural network model is demonstrated to accurately predict the unsteady turbine performance given a limited number of training data. Lastly, a gradient-based optimization method is developed to identify the optimum working conditions, in terms of pulse shape, to maximize the power output of the turbine.

High-fidelity LES simulations are used to improve the understanding of flow physics. The flow at the rotor blade experiences different characteristics between continuous and pulsating flow conditions. In particular, large separations and secondary flows develop on both the pressure and suction sides of the blade as
a consequence of the large range of relative inflow angles the blade is exposed to. Such secondary flows are addressed as the main cause of the drop of the isentropic efficiency from continuous to pulsating flow conditions.

**Key words:** Turbocharger, Radial Turbine, Pulsating Flow.
Sammanfattning


LES simuleringar med hög noggrannhet används för att förbättra förståelsen av flödesmekaniken i turboaggregatet. Flödet vid rotorblandet utsätts för olika inloppsvillkor under stabila test-villkor och under pulserande flödesförhållanden.
I synnerhet utvecklas stora separationer av flödet och sekundära flödesstrukturer på både tryck- och sugsidan av bladet. Detta till följd av det stora området av relative inflödesvinklar som bladet utsätts för. Sådana sekundära flödesstrukturer ses som huvudsaken till reduceringen av den isentropisk verkningsgraden från stabila test-villkor till pulserande flödesförhållanden.

Nyckelord: Turboladdare, Radiell Turbin, Pulserande Flöde.
Preface

This thesis deals with the pulsating flow in turbocharger radial turbines. A brief introduction on the basic concepts and methods is presented in the first part. The second part contains seven articles. The papers are adjusted to comply with the present thesis format for consistency, but their contents have not been altered as compared with their original counterparts.


September 2022, Stockholm

*Roberto Mosca*
Division of work between authors

The main advisor for the project is Mihai Mihaescu (MM). Anders Dahlkild (AD) acts as co-advisor.

**Paper 1.** The set-up of the simulations has been done by Roberto Mosca (RM). The simulations and analysis were done by RM. The paper was written by RM with comments and feedback from Mihai Mihaescu (MM).

**Paper 2.** The set-up of the simulations has been done by RM. The simulations and analysis were done by RM. The paper was written by RM with comments and feedback from Shyang Maw Lim (SML) and MM.

**Paper 3.** The set-up of the simulations has been done by RM. The simulations and analysis were done by RM. The paper was written by RM with comments and feedback from MM.

**Paper 4.** The set-up of the simulations has been done by RM. The simulations and analysis were done by RM. The paper was written by RM and Marco Laudato (ML) with comments and feedback from ML and MM.

**Paper 5.** The set-up of the simulations has been done by RM. The simulations and analysis were done by RM. The paper was written by RM and ML with comments and feedback from ML and MM.

**Paper 6.** The set-up of the simulations has been done by RM and SML. The simulations and analysis were done by RM. The paper was written by RM with comments and feedback from SML and MM.

**Paper 7.** The set-up of the simulations has been done by RM and SML. The simulations and analysis were done by RM. The paper was written by RM with comments and feedback from SML and MM.

Conferences

Part of the work in this thesis has been presented at the following international conferences. The presenting author is underlined.


# Contents

Abstract iii  
Sammanfattning v  
Preface vii  

Part I - Overview and summary

Chapter 1. Introduction 1  
  1.1. Overview on greenhouse gas emissions 1  
  1.2. The role of turbocharger 2  
  1.3. Pulsating flow 4  
  1.4. Heat transfer 9  
  1.5. Objectives and motivations 10  

Chapter 2. Theory of Radial Turbines 12  
  2.1. Quasi-steady performance 12  
  2.2. Pulsating flow performance 14  
  2.3. Computational model 14  

Chapter 3. Governing Equations 17  
  3.1. Compressible flow equations 17  
  3.2. The nature of turbulence 18  
  3.3. Reynolds-averaged Navier-Stokes equations 20  
  3.4. Large eddy simulations 23  
  3.5. Boundary layer theory 24  
  3.6. CFD for rotating machinery 24  
  3.7. Numerical methods 25  

Chapter 4. Summary of results 28  
  4.1. Model validation 28
4.2. Flow analysis 33
4.3. Performance analysis 39

Chapter 5. Conclusions and outlook 60

Acknowledgements 64

Bibliography 66
Part II - Papers

Summary of the papers

Paper 1. Analysis of the volute-rotor interaction in radial turbines by means of large eddy simulations

Paper 2. Large eddy simulations of a turbocharger radial turbine under pulsating flow conditions

Paper 3. Assessment of the unsteady performance of a turbocharger radial turbine under pulsating flow conditions: parametric study and modeling


Paper 5. Gradient-based optimization of pulsating inflow conditions for turbocharger radial turbines

Paper 6. Turbocharger radial turbine response to pulse amplitude

Paper 7. Influence of pulse characteristics on turbocharger radial turbine
Part I

Overview and summary
Chapter 1

Introduction

Fluid machines describe a wide variety of machines that transfer energy between a rotor and a working fluid. They are firstly grouped into two categories based on the direction of the energy transfer. In turbines, the transfer of energy is directed from the working fluid to the rotor. Conversely, in compressors or pumps, the energy transfer is directed from the rotor to the fluid. Turbomachinery holds a central role in the development of a more equal and sustainable society since it is at the basis of a large variety of fundamental processes of modern society, such as electricity production in hydro-electric, steam, and wind power plants, and road, marine, and aviation transportation. Due to the large variety of processes involved, the development and improvement of more efficient turbomachines are fundamental processes to face the rapidly growing needs of our energy-consumptive society.

1.1. Overview on greenhouse gas emissions

Global warming is the result of the increasing concentration of greenhouse gases in the atmosphere due to anthropogenic activities. According to the United States Environmental Protection Agency EPA (2022), the most dangerous greenhouse gases are represented by Carbon Dioxide (CO$_2$), Methane (CH$_4$), and Nitrous Oxide (N$_2$O) which are emitted in different proportions in the atmosphere (see fig.1.1 for the distribution of the greenhouse gases in the U.S.). Carbon dioxide represents the vast majority of greenhouse gas emissions in both

![Figure 1.1: Total U.S. greenhouse gas emissions in 2020.](image)
U.S. and Europe and its emissions are the result of chemical processes involving burning fossil fuels and biological materials. To understand the actions that different countries have adopted to reduce CO₂ emissions in the atmosphere, it is interesting to first analyze the primary sources of carbon dioxide emissions. According to Fig.1.2 from EPA (2022), 33% of carbon dioxide emissions are produced by the transportation sector, followed by the electricity production (31%), and industry (16%). More deeply in the transportation sector, light-duty vehicles account for 58% of the greenhouse gas emissions, followed by heavy-duty vehicles (29%), and aircraft transportation (10%). Based on the dominant position of the transportation sector in the CO₂ emissions, countries have started to introduce stringent regulations in terms of fuel consumption and greenhouse gas emissions. In Europe, new regulations established by the European Parliament and Council (2019) targeting a reduction of 15% and 37.5% of CO₂ emissions by 2025 and 2030 have boosted the development of new technologies such as hybrid, electric, fuel cells, and hydrogen vehicles. According to the data published by the European Automobile Manufacturers’ Association ACEA (2022), vehicle market has reacted to the new regulations with a transition from a market completely dominated by petrol and diesel vehicles in 2018 (93%) to a more diversified one (see Fig.1.3). However, the petrol and diesel vehicles still represent the largest fraction of the vehicles sold with a percentage equal to 60% in 2021.

1.2. The role of turbocharger
Among the different strategies available, engine downsizing combined with boosting technologies has established as the most effective gas emission reduction technology available for both gasoline and diesel ICEs (Alshammari et al. 2019; Namar et al. 2021). The idea is based on reducing the frictional and thermal losses within the system by reducing the size of the system itself. To understand the impact of engine downsizing and the consequent need for boosting technologies, it is useful to consider the simplified model for the power generated by the engine proposed by Heywood (2018) (see Eq.1.1).
1.2. The role of turbocharger

\[ \dot{W} = C \cdot n \cdot p_{me} \cdot V_{sw} \]  

(1.1)

According to Eq.1.1, the engine power \( \dot{W} \) is primarily determined by the rotational speed of the engine \( n \), the break mean effective pressure \( p_{me} \), and the swept cylinder volume \( V_{sw} \). \( C \) is a dimensionless constant equal to 1 for two-stroke engine type or equal to \( 1/2 \) for four-stroke engine type. All these parameters differently contribute to the internal mechanical losses. The engine speed is associated with a quadratic increase of the friction losses, while the losses show no variation with respect to the break mean effective pressure and a linear relation with respect to the swept cylinder volume (Leduc et al. 2003; Guzzella et al. 2007; Ben-Chaim et al. 2013). According to Eq.1.1, reducing the size of the engine, i.e. the swept cylinder volume, has the effect of decreasing the power outcome together with lower frictional and thermal losses due to the smaller size of the system. As a consequence, to maintain the same level of engine power, it is necessary to increase the break mean effective pressure by increasing the density of the air filling the cylinders of the engine. This is generally achieved in supercharged and turbocharged engines by means of a radial compressor positioned upstream to the cylinder engine. The way the compressor is driven distinguishes supercharging from turbocharging. In the first case, the rotational motion of the compressor is induced by the engine crankshaft, which the compressor can be connected through a belt, shaft or chain. In turbocharged systems, the necessary kinetic energy to spin the compressor is extracted from the exhaust gases by a radial turbine, which is connected through the same shaft to the compressor (see Fig.1.4 for a sketch). Turbocharged or supercharged systems can be used in two different ways: increasing the power while maintaining the same \( V_{sw} \), or maintaining the same power while decreasing \( V_{sw} \). In the optic of reducing losses, the second option is chosen. As a consequence, downsized engines are characterized by a swept cylinder volume that is 30 to 50% smaller than naturally aspirated engines, (Ricardo et al. 2011; Subramani & Ramesh 2022).

Figure 1.3: Fuel type cars sold in Europe in years 2018 and 2021, respectively.
1. Introduction

Figure 1.4: Turbocharger set-up (courtesy of Garrett).

In Fig. 1.4 a schematic sketch for a turbocharger is reported. The radial turbine is connected through a series of bent pipes to the engine cylinders. The exhaust gases are discharged after the combustion process by the reciprocating cylinder and are driven by the exhaust manifold to the radial turbine. Here, the exhaust gases expand across the turbine, which converts the kinetic energy of the fluid in rotational energy driving the rotation of the compressor. On the compressor side, cold ambient air is sucked into the compressor through the intake pipe and is guided through the volute to the cylinders of the engine.

1.3. Pulsating flow

Radial turbines for turbocharger applications are a unique example in the turbomachinery field due to the particular inflow conditions they are exposed to. Indeed, under on-engine conditions, the turbine system is fed by a pulsating flow caused by the reciprocating pistons of the engine. Pulsating flow is defined as the periodic oscillation on a steady stream (Esfe et al. 2021; Ye et al. 2021). The interest in such kind of flows is large, since it is encountered in many natural and industrial applications, such as blood flows in veins, internal combustion engines, coolers and pulse jet engines (Esfe et al. 2021). As stated by Wang & Zhang (2005), the literature distinguishes between pulsating (or pulsatile) flows, when the time-averaged velocity is non-zero, and oscillating (or oscillatory) flows, when the time-averaged velocity is zero (see Fig. 1.5). In pulsating flow applications, to facilitate the applicability of the results, three different non-dimensional numbers are used. For a signal in the form of $u = u_s + A \sin(\omega t)$, with $u_s$ the steady component of the velocity, $A$ the amplitude of the oscillation, and $\omega = 2\pi f$ the angular frequency, two different Reynolds numbers are defined: the first for the stable component of the velocity (see Eq. 1.2), the latter for the oscillating components (see Eq. 1.3), respectively.

$$Re_s = \frac{u_s L}{\nu} \quad (1.2)$$
1.3. Pulsating flow

\[ Re_\omega = \frac{A^2}{\nu \omega} \] (1.3)

Here, \( L \) is defined as the characteristic length of the problem investigated and \( \nu \) represents the kinematic viscosity of the fluid. The third non-dimensional number used to describe the effects of the oscillatory component in pulsating flows is the Womersley number, identified by the nomenclature \( Wo \) or \( \alpha \). It represents the ratio between the transient inertial forces and the viscous forces (see Eq.1.4).

\[ \alpha^2 = \frac{\text{transient inertial forces}}{\text{viscous forces}} = \frac{\omega L^2}{\nu} \] (1.4)

When the Womersley number is small (\( \alpha \ll 1 \)), the viscous effects dominate and the velocity assumes the characteristic parabolic profiles typical of laminar flows. Variations of the velocity are in phase with the derivative of the pressure and, in case of pipe flows, Poiseuille’s law represents a good approximation. For large values of the Womersley number (\( \alpha \gg 10 \)), the pulsation effects become significant, the velocity assumes a flat profile, and the velocity variations are out of phase with the pressure gradient by approximately \( \pi/2 \). In turbocharger applications, the frequency of the pulse ranges from 20 Hz to 200 Hz, so that \( \alpha \) can always be considered much larger than 10 under on-engine conditions.

The pulsating flow that feeds the hot-side exposes the turbocharger turbine to a wide range of operating conditions determining also performance deviations from quasi-steady performance. As a consequence, the quasi-steady assumption, which is based on the argument that the pulse frequency is much lower than the blade passing frequency, cannot be always applied to describe the turbine behavior in pulsating flow conditions. The instantaneous performance enclose

Figure 1.5: Examples of pulsating (Fig.1.5a) and oscillating (Fig.1.5b) flows.
the quasi-steady ones into the so-called hysteresis loop (see Fig.1.6 for an example) that is generally represented as mass flow parameter (MFP) against the expansion ratio ($\pi$) or isentropic efficiency ($\eta_{is}$) against blade speed ratio (BSR). The area enclosed by the loop, a measure of the intensity of the unsteadiness, is significantly affected by the pulse characteristics such as amplitude and frequency. As established by numerous experimental campaigns (Szymko et al. 2005; Copeland et al. 2011; Piscaglia et al. 2019) and numerical investigations (Galindo et al. 2013; Yang et al. 2016; Zhao et al. 2018; Mosca & Mihaescu 2022), increasing the pulse amplitude and frequency has the effect of enlarging the area encapsulated by the hysteresis loop. A third pulse characteristic that is demonstrated to affect the turbine unsteady behavior is the time derivative of the pressure pulse (or temporal gradient). Cao et al. (2014) show that, for a steeper slope of the pressure pulse, the effects of unsteadiness intensify, while Mosca et al. (2022) register a 3.2% decrease of the turbine power for a 10% reduction of the mass flow rate temporal gradient. Moreover, Marelli & Capobianco (2011) demonstrate that the size of the hysteresis is also proportional to the waste gate angle of aperture in an experimental campaign.

The consensus on the nature of the turbine unsteady behavior is based on the idea that performance deviations are caused by the internal volumes of the hot-side system (Szymko et al. 2005). Indeed, during pulsating flow conditions, the measured inflow conditions do not reflect the actual state of the flow inside the rotor due to the spatial distance between the measurement location and the rotor shaft. Due to the compact nature of the system, it is rather difficult to experimentally obtain measurements close to the rotor shaft, so the major efforts to validate this idea come from numerical simulations. Cao et al. (2014), in a numerical study of a radial rotor without volute, show that, by correcting the phase shift by a combination of bulk flow velocity and sonic speed, the hysteresis loop collapses on the quasi-steady performance line. Other authors demonstrate that moving the inlet measurement location closer to the turbine

---

![Figure 1.6: Example of hysteresis loop as mass flow parameter (MFP) against expansion ratio ($\pi$).](image-url)
inlet section has the effect of reducing the intensity of the unsteadiness since this approach reduces the particle path and, consequently, the residence time (Galindo et al. 2013; Liu & Copeland 2018; Mosca & Mihaescu 2022).

In recent years, different non-dimensional numbers have populated the literature with the aim of predicting whether the turbine operates as a quasi-steady device or not. At first, Chen & Winterbone (1990) and Chen et al. (1996) made use of the Strouhal number (see Eq.1.5).

\[ St = \frac{fL}{v} = \frac{t_f}{t_c} \]  

(1.5)

Here, \( f \), \( L \), and \( v \) define the characteristic frequency, length, and velocity, respectively. Alternatively, the ratio between \( L \) and \( v \) can be expressed as the residence time of a fluid particle \( t_f \). The Strouhal number expresses the importance of the mass rate of change inside a volume relative to the mass flux in the continuity equation. As the time for a fluid particle to travel through the volume increases with respect to the time period of the pulse, the mass accumulation becomes significant. Since the Strouhal number is directly linked to the characteristic length \( L \), the volute has a great impact on the mass accumulation, since it represents the largest volume of the system. Starting from the Strouhal number, different and more complex non-dimensional numbers have been introduced. Szymko et al. (2005) introduced the modified version of the Strouhal number (see Eq.1.6) normalized by the pulse length fraction \( \phi \).

\[ MSt = \frac{1}{2\phi} \frac{fL}{v} \]  

(1.6)

For values of \( MSt \) larger than 0.1 the unsteady effects are significant. Subsequently, Copeland et al. (2012a) introduced the Lambda (\( \Lambda \)) criterion to account for the effects of the pulse amplitude.

\[ \Lambda = \Pi \cdot St = \frac{2A}{\gamma \bar{p}} \frac{fL}{v} \]  

(1.7)

In Eq.1.7, \( A \) represents the pulse amplitude, \( \gamma \) the heat capacity ratio, and \( \bar{p} \) the average pressure of the pulse. If \( \Lambda \) is larger than unity, the mass rate of change cannot be neglected and leads to a discrepancy of conditions between the inlet and outlet of the volume. The \( St \), \( MSt \), and \( \Lambda \) criteria describe the turbine unsteady behavior in terms of average performance. To overcome this limitation, Cao et al. (2014) introduced the normalized reduced frequency (see Eq.1.8) to account for the instantaneous pressure variation effects.

\[ |\varepsilon(t)| \beta(t) = \frac{|\Delta p|}{\bar{p}} \frac{t_f}{\Delta t} \]  

(1.8)

Here, \( \Delta p \) represents the pressure variation in a certain time interval \( \Delta t \), \( t_f \) the residence time of a fluid particle, and \( \bar{p} \) the average value of the pressure pulse. The main advantage of the \( |\varepsilon(t)| \beta(t) \) non-dimensional number is the ability
to detect when the turbine operates as a quasi steady-device instantaneously, and not in average. According to Cao et al. (2014), a value of 0.07 identifies the threshold between quasi-steady and unsteady behavior. Interestingly, by averaging $|\varepsilon(t)|/|\beta(t)|$ in time and dividing it by the heat capacity ratio $\gamma$, Eq.1.8 reduces to the $\Lambda$ criterion. Moreover, Eq.1.8 does not depend on the pulse amplitude or frequency, but only on the time derivative of the pressure pulse and residence time.

Understanding the behavior of the turbine response to the pulse shape is important not only to model its performance but to optimize the turbine working conditions as well. Up to now, most of the works on the turbine unsteady performance have focused on the characterization of the performance deviations from quasi-steady to unsteady flow conditions. Few other works, summarized in Tab.1.1, attempt to highlight characteristic trends of the turbine performance through independent and discrete variations of the pulse amplitude and frequency. The difficulties in the analysis and comparison of such works are represented by the uncertainties of boundary conditions. In experimental applications, uncertainties are related to the difficulties in controlling the pulse shape, i.e. decoupling the effects of pulse amplitude and frequency. In CFD applications, the uncertainties are related to the method adopted to vary the pulse characteristics. In numerical works, where total conditions (total pressure $p_t$ and total temperature $T_t$) are generally imposed at the inlet boundary, the method of varying the total pressure pulse to the desired amplitude while the respective total temperature pulse is derived according to the isentropic relation $p_t^{1-\gamma}T_t^\gamma = \text{const}$ is the most common. However, the validity of this hypothesis has never been discussed according to the author’s knowledge and may be questionable. Moreover, the third factor of uncertainty is related to the different energy content in the flow as pulse characteristics are changed. As a consequence, the response of the turbine performance to variations of the pulse amplitude and frequency does not show homogeneous trends across the literature (see Tab.1.1). Szynko et al. (2005) register an increase of the isentropic efficiency by increasing the pulse frequency from 40 Hz to 100 Hz, while Rajoo et al. (2012) report a non-monotonic trend on a twin-entry variable geometry. In a numerical study of a two-stage turbocharger system, Zhao et al. (2018) register a decrease in the high-pressure turbine isentropic efficiency with increasing pulse amplitude and frequency. Mosca et al. (2022b) relate the efficiency drop to the growth of the total internal irreversibilities caused by larger viscous dissipation as the pulse amplitude increases. The shape of the pulse is another important factor that determines the turbine performance. In particular, Lee et al. (2018) and Rezk et al. (2021) found that a radial turbine subject to a square pulse is characterized by a smaller isentropic efficiency as compared to a triangular, sinusoidal and realistic pulse.
Table 1.1: Summary of the works focusing on the turbine performance response to pulse amplitude and frequency variations. Characteristic trends of the turbine performance, expressed by the expansion ratio ($\pi$), isentropic efficiency ($\eta_{is}$), and turbine torque ($\tau$), are reported for increasing values of pulse amplitude and frequency. Upwards and downwards arrows denote the increase or decrease of the quantity analyzed, while the double arrow indicates a non-monotonic behavior.

<table>
<thead>
<tr>
<th>Reference</th>
<th>Setup</th>
<th>Turbine Model</th>
<th>Nozzle</th>
<th>$\pi$</th>
<th>$\eta_{is}$</th>
<th>$\tau$</th>
<th>$\pi$</th>
<th>$\eta_{is}$</th>
<th>$\tau$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Karamanis et al. (2001)</td>
<td>Exp.</td>
<td>Mixed Flow</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Szymko et al. (2005)</td>
<td>Exp.</td>
<td>Mixed Flow</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Copeland et al. (2011)</td>
<td>Exp.</td>
<td>Double Entry</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Rajoo et al. (2012)</td>
<td>Exp.</td>
<td>Twin Entry</td>
<td>Yes</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Padzillah et al. (2014)</td>
<td>Sim.</td>
<td>Single Entry</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Yang et al. (2015)</td>
<td>Sim.</td>
<td>Single-Entry</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Zhao et al. (2018)</td>
<td>Sim.</td>
<td>Two-Stage</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Zhao et al. (2019)</td>
<td>Sim.</td>
<td>Two-Stage</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Piscaglia et al. (2019)</td>
<td>Exp.</td>
<td>Single Entry</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Rezk et al. (2021)</td>
<td>Sim.</td>
<td>Single Entry</td>
<td>Yes</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Mosca et al. (2022)</td>
<td>Sim.</td>
<td>Single Entry</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Mosca et al. (2022b)</td>
<td>Sim.</td>
<td>Single Entry</td>
<td>No</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
</tbody>
</table>

1.4. Heat transfer

Under on-engine conditions, the temperature of the exhaust gases feeding the turbine may be greater than 1000 K and significant thermal processes affect the turbine and, consequently, the entire turbocharger behavior. As reported by Burke et al. (2015), due to the compact nature of the system and the high temperatures of the exhaust gases, heat transfer occurs from the turbine to the compressor and the surroundings and may account for more than 20% of the total turbine power. As a consequence, the turbocharger operates as a diabatic machine and the enthalpy difference across the turbine cannot be used to calculate the work extracted by the turbine. Rautenberg et al. (1984), Rautenberg & Kammer (1984), and Malobabic & Rautenberg (1987) are among the pioneering researchers that rose the question on the effects of the heat transfer on turbochargers. It is essential to account for heat transfer effects to simulate reliable performance maps and improve turbocharger-engine matching. Indeed, since the heat transfer causes the temperature to be higher at the compressor outlet and lower at the turbine outlet, the temperature measurements cannot be used to calculate the power supplied or extracted. The higher temperature at the compressor outlet would be accounted for as an apparent increase in the power consumption, decreasing the compressor isentropic efficiency. Conversely, since the turbine power is inferred at the compressor stage, the turbine efficiency would appear larger. As a result, the compressor performance is underestimated while the turbine one is overestimated, with the possibility for the isentropic efficiency
to reach nonphysical values (Romagnoli et al. 2017). Different models accounting for the heat transfer effects have been proposed in the literature to predict the diabatic efficiency. The main differences rely on the authors’ assumptions about whether heat transfer occurs before or after the compression/expansion process (Sirakov & Casey 2013; Romagnoli et al. 2017). The apparent effect influences the isentropic efficiency particularly at the lower characteristic speeds, where the difference is up to 20% for both compressor and turbine, see e.g. Sirakov & Casey (2013). Nowadays, several authors have shown the detrimental impact of heat transfer on rotor efficiency, especially at low rotational speeds. Romagnoli & Martinez-Botas (2012) reported an efficiency reduction up to 30% from adiabatic to diabatic conditions. In an experimental campaign, Shaaban & Seume (2012) registered a 55% turbine power decrease compared to adiabatic conditions and demonstrated the effectiveness of thermal insulation to increase the turbine power. On the numerical side, Lim et al. (2018) reported a decrease of the turbine power up to 10%, for increasing levels of heat transfer, compared to adiabatic conditions.

1.5. Objectives and motivations

This thesis is primarily motivated by the need of enhancing the understanding of the pulsating flow effects on the performance of radial turbines for turbocharger applications. This is achieved by the utilization of different CFD approaches (LES) or models (RANS-based formulations) chosen concerning the specific target. From a fluid physics point of view, each component of the hot-side represents a fundamental problem on its own. In Paper 1, Large Eddy Simulations are used to study the effects of the volute tongue on the flow distribution around the volute-rotor interface. It is found that the volute tongue wakes generate disturbances in the turbine inflow conditions that significantly affect the blade performance. In Paper 2, Large Eddy Simulations are used to characterize the secondary flows developing during pulsating flow conditions. Under pulsating flow conditions, the turbine blade is subject to a wide range of inflow conditions in terms of incoming flow direction. At low mass flow rate conditions, the blade is exposed to large negative relative inflow angles which determine large separation zones on the pressure side. As the mass flow rate increases, the separation on the pressure side gradually reduces and moves to the suction side. In Paper 3, a more classical investigation of the unsteady turbine performance is carried out using unsteady Reynolds-averaged Navier-Stokes (RANS) with respect to pulse amplitude and frequency variations. This study aims to highlight characteristic trends of the turbine performance for pulse amplitude and frequency variations. At the end of the paper, an algebraic model to predict the intensity of the deviations from quasi-steady performance is proposed. The model highlights the time derivative of the pressure pulse as the central parameter influencing the intensity of unsteadiness. In Paper 4, a data-driven method is developed and employed to predict the turbine performance under unsteady flow conditions. The data set derived from Paper 3 is used to train a Fully-Connected Neural Network (FCNN) to predict the
instantaneous turbine torque and relative inflow angle, given a certain pulse shape as input condition. This approach represents an accurate and efficient method to model radial turbine performance under pulsating flow conditions. In **Paper 5**, the optimum working conditions, in terms of pulse shape, are defined through a gradient-based method. The algorithm exploits the NN model developed in **Paper 4** to efficiently calculate the cost function, which is chosen as the time-averaged turbine torque. In the paper, the effects of different constraints on the optimal pulse shape are shown. Latly, in **Paper 6** and **Paper 7**, an exergy-based approach is used to characterize the aerothermodynamic losses in the hot-side system, including the exhaust manifold, with respect to variations of the pulse shape. To overcome uncertainties related to the change of the gas energy as the pulse shape is modified, a constraint on the inflow exergy is applied when designing the pulses used as boundary condition. In these works, specific trends of the aerothermodynamic losses are highlighted with respect to pulse amplitude, frequency, and temporal gradient variations.
Chapter 2

Theory of Radial Turbines

In a turbocharger system, the necessary work to spin the compressor is extracted from the exhaust gases using a radial turbine connected through a common shaft. At the turbine inlet, the flow is smoothly oriented perpendicularly to the rotor shaft in the volute and is then redirected in the axial direction at the rotor exit after the expansion. In Fig. 2.1, a schematic sketch of the inlet velocity triangles is illustrated. In the figure, the velocity components $c$, $c_m$, and $c_\theta$ represent the absolute velocity vector and its components in the radial and tangential direction. $u$ indicates the blade tip velocity, while $w$ and $w_\theta$ are the velocity vector and its tangential component in the reference frame of rotation. Based on the conservation of angular momentum, the turbine power can be calculated as in Eq. 2.1.

$$\dot{W}_T = \tau \omega = \dot{m} (u_1 c_\theta 1 - u_2 c_\theta 2)$$

(2.1)

Here, $\omega$ represents the angular velocity of the rotor in rad/s, $\tau$ is the turbine torque, and the subscripts 1 and 2 denote the inlet and outlet of the turbine, respectively. Now, by using the second law of thermodynamics ($\dot{Q} - \dot{W} = \dot{m}\Delta h_t$ with $\Delta h_t$ the total enthalpy change) and by assuming adiabatic boundary conditions, one can derive the Euler turbine equation (see Eq. 2.2).

$$\dot{W}_T = \dot{m} \, c_p \,(T_{i1} - T_{i2}) = \dot{m} \,(u_1 c_\theta 1 - u_2 c_\theta 2)$$

(2.2)

As mentioned in Section 1.4, since the hot-side operates in diabatic conditions during pulsating flow, the change of total enthalpy is not applicable to calculate the power extracted by the turbine since, under pulsating flow conditions, the heat transfer rate $\dot{Q}$ is generally non-zero.

2.1. Quasi-steady performance

The performance of a turbocharger are measured experimentally in hot continuous flow conditions in specific gas-stand facilities. The most common set-up is represented by the two-loop circuit type that is characterized by two different independent circuits, one for the compressor and one for the turbine, respectively. This type of facility is used to characterize the turbocharger behavior by independently reporting the compressor and turbine performance into the
2.1. Quasi-steady performance

so-called performance maps. Examples of performance map for the turbine are given in Fig.2.2a and 2.2b as mass flow parameter (MFP) against expansion ratio ($\pi$) and combined isentropic and mechanical efficiency ($\eta_{is} \times \eta_m$) against expansion ratio, respectively, for different speed lines. The mass flow parameter (see Eq.2.3) is a pseudo non-dimensional number that characterizes the conditions of the flow at the inlet.

$$\text{MFP} = \frac{\dot{m}\sqrt{T_t}}{p_t} \quad (2.3)$$

$m$, $T_t$, and $p_t$ represent the mass flow rate, total temperature, and total pressure measured at an inlet location upstream of the turbine. The expansion ratio (see Eq.2.4) and isentropic efficiency (see Eq.2.5) are two indices of the turbine performance. The expansion ratio expresses the expansion of the flow as the ratio between the total pressure measured upstream to the turbine and the static pressure measured downstream of the turbine. The isentropic efficiency measures how far the turbine operates from an ideal isentropic expansion and is defined as the ratio between the extracted and isentropic power. In Eq.2.5, $c_p$ and $\gamma$ represent the specific heat at constant pressure and the heat capacity ratio, respectively.

$$\pi = \frac{p_{t,\text{in}}}{p_{\text{out}}} \quad (2.4)$$

$$\eta_{is} = \frac{\dot{W}_T}{\dot{W}_{is}} = \frac{\tau \omega}{\dot{m} c_p T_t \left(1 - \pi \frac{\gamma - 1}{\gamma}\right)} \quad (2.5)$$

In Fig.2.2b, the combined isentropic and mechanical efficiency is used. The mechanical efficiency (see Eq.2.6) is defined as the ratio between the compressor power ($\dot{W}_C$) and turbine power ($\dot{W}_T$), which is equal to the compressor power plus a power loss term ($\dot{W}_f$) due to the friction losses generally associated to the bearing system.

$$\eta_m = \frac{\dot{W}_C}{\dot{W}_T} = \frac{\tau \omega}{\dot{m} c_p T_t \left(1 - \pi \frac{\gamma - 1}{\gamma}\right)} + \frac{\dot{W}_f}{\dot{W}_T}$$

Figure 2.1: Schematic sketch of the inlet velocity triangles.
2. Theory of Radial Turbines

\[ \eta_{is} = \frac{\int_0^{t_p} \tau \omega \, dt}{\int_0^{t_p} \dot{m}_c \, c_p \, \bar{T}_i \left(1 - \pi^{\frac{1}{\gamma}}\right) \, dt} \]  

2.2. Pulsating flow performance

The behavior of a turbine is generally represented by time-averaged performance indices. However, under pulsating flow conditions, the phase shift that occurs between the physical quantities measured at the inlet of the system and the turbine torque measured at the rotor shaft must be taken into account. As a consequence, the time-averaged isentropic efficiency (see Eq.2.7) is calculated as the ratio between the time integrals of the turbine and isentropic power.

\[ \eta_m = \frac{\dot{W}_C}{\dot{W}_T} = \frac{\dot{W}_C}{(\dot{W}_C + \dot{W}_f)} \]  

2.3. Computational model

The system investigated in the present thesis is constituted by a single-entry 12-blade turbocharger turbine applied in a 4-cylinder engine for passenger cars. In the thesis, two different configurations of the model are used. In the first one (system A in Fig.2.3), the volute is preceded by a straight pipe 5.5 volute.
diameters long while, in the second configuration (system B in Fig. 2.4), the complex system of bent pipes connecting the turbine system to the engine is maintained. The second configuration of the system represents a more realistic scenario since it accounts for the secondary flows developing inside the exhaust manifold as demonstrated by Lim et al. (2019). However, the inclusion of
the exhaust manifold significantly increases the computational cost since, to complete an entire engine cycle, four pulses must be simulated. This aspect increases the computational cost by four times as compared to system A and is the reason why most numerical works only consider a simplified pipe preceding the volute. In both computational models, the rotating region is followed by a diffuser and a convergent duct. This is meant to reduce the effects of the reverse flow at the outlet section. Downstream of the turbine, the outlet section is placed at 13 volute diameters from the rotor-diffuser interface ($S_3$ with reference to fig.2.3) to minimize the influence of the imposed boundary conditions on the predicted flow field. System A is used in Paper 1-5, while the influence of the exhaust manifold is studied in Paper 6 and Paper 7 using system B.
Chapter 3

Governing Equations

Fluid, like all matter, is constituted by molecules separated by empty space (Katopodes 2019). However, when the characteristic length of the problem analyzed is orders of magnitudes larger than the molecular scales, the fluid can be considered a continuum. As a consequence, the properties of the fluid, such as density, pressure, and temperature, are considered uniform within fluid particles. The continuum hypothesis is assumed valid when the Knudsen number,

\[ Kn = \frac{\lambda}{L} \]  (3.1)

defined as the ratio between the molecular mean free path and the characteristic length of the problem, is smaller than 0.01. Since the order of the mean free path of air in ambient conditions is \(10^{-7}\) m and the order of the characteristic length, assuming the diameter of the inlet pipe, is \(10^{-2}\) m, the continuum hypothesis can be considered valid for the flow in a turbocharger turbine.

3.1. Compressible flow equations

The governing equations, stating the conservation of mass, momentum, and total energy, are given by the compressible flow equations expressed in Cartesian coordinates in Eq.3.2-3.4.

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} = 0 \]  (3.2)

\[ \frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} + \rho f_i \]  (3.3)

\[ \frac{\partial (p e_i)}{\partial t} + \frac{\partial (\rho u_j e_i)}{\partial x_j} = -\frac{\partial (p u_j)}{\partial x_j} - \frac{\partial q_j}{\partial x_j} + \frac{\partial (u_i \sigma_{ij})}{\partial x_j} \]  (3.4)

Here, \(\rho\), \(u_i\), and \(p\) represent the fluid density, the absolute velocity in the \(x_i\) direction in Cartesian coordinates, and the absolute static pressure, respectively. \(\sigma_{ij}\) represents the viscous stress tensor while the term \(f_i\) in the momentum equation is the sum of body and external forces, if present. The total energy per unit mass is defined as \(e_t = e + \frac{1}{2} u_i u_i\) with the specific energy defined as \(e = c_v T\).
3. Governing Equations

The system of equations, as presented, is not determined since the number of unknowns is larger than the number of equations. As a consequence, assumptions on the physics of the flow must be provided as constitutive relations. Since the fluid is considered as Newtonian, the viscous stress tensor \( \sigma_{ij} \) linearly depends on the strain rate tensor \( S_{ij} \) as in Eq.3.5.

\[
\sigma_{ij} = 2\mu S_{ij} - \frac{2}{3} \mu S_{kk} \delta_{ij} \tag{3.5}
\]

The strain rate tensor \( S_{ij} \) represents the symmetric part of the velocity gradient tensor and reads as

\[
S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{3.6}
\]

The ideal gas assumption is used as second constitutive relation in order to link pressure and temperature by the ideal gas law as in Eq.3.7, where \( R \) represents the ideal gas constant \( (c_p = c_v + R) \).

\[
\frac{p}{\rho} = RT \tag{3.7}
\]

In Eq.3.4, the total energy rate of change is equal to the heat flux \( q_i \) and a contribution given by the shear stress tensor. The heat flux is modeled according to Fourier’s law (see Eq.3.8),

\[
q_i = -k \frac{\partial T}{\partial x_i} \tag{3.8}
\]

where \( k \) represents the thermal conductivity coefficient and \( T \) is the static temperature. Given the constitutive relations above mentioned, the system of governing equations is now determined and can be solved.

Under on-engine conditions, the flow field inside the hot-side is characterized by an intense heat transfer as a result of the high temperatures of the gas \( (T > 1000 \text{ K}) \). As a consequence of the large temperature variations, the dynamic viscosity \( \mu \) cannot be assumed constant. For this reason, the dynamic molecular viscosity is modeled by Sutherland’s law (see Eq.3.9).

\[
\frac{\mu}{\mu_0} = \left( \frac{T}{T_0} \right)^{\frac{3}{2}} \left( \frac{T_0 + T_s}{T + T_s} \right) \tag{3.9}
\]

Here, \( \mu_0 \) represents the molecular dynamic viscosity at the reference temperature \( T_0 \) and \( T_s \) Sutherland’s constant depending on the type of gas considered which, in the present case, is air.

3.2. The nature of turbulence

Under certain conditions, the solution of the Navier-Stokes equations exhibits chaotic behavior. This aspect is shown in the very famous experiment by
Reynolds (1883), where dye is injected to follow the trajectories of the particles in a water pipe flow for different flow velocities. The experiment shows that, at low velocities, the dye particles exhibit a very regular and laminar behavior. However, when the velocity of the flow is increased, the filament of dye breaks up due to the chaotic motion of the flow. This phenomenon is the result of the unbalanced between the inertial and the viscous forces that is expressed by the non-dimensional number in Eq.3.10, named in honor of Reynolds himself.

\[ Re = \frac{\rho U L}{\mu} \]  

(3.10)

The Reynolds number expresses the ratio between the inertial and viscous forces. Here, \( \rho \) represents the density of the fluid, \( U \) the characteristic velocity, \( L \) the characteristic length, and \( \mu \) the dynamic viscosity of the fluid.

The first qualitative attempt to characterize the behavior of turbulence is due to Richardson (1922). He observed that turbulence is composed of eddies of different sizes. The large eddies, characterized by high Reynolds numbers and unaffected by viscous forces, are unstable and break up into smaller structures. This process continues until the equilibrium between inertial and viscous forces is reached and the kinetic energy is dissipated by the molecular viscosity into heat. The process of transfer of energy from large scales to smaller scales is referred to as energy cascade and inspired Kolmogorov (1941) and Obukhov (1941) to develop a more quantitative description of turbulence. In his first hypothesis, Kolmogorov states that the small scales are statistically isotropic at sufficiently high Reynolds numbers. As a consequence, the geometrical information contained in the large scales is lost at the small scales, which exhibit a universal behavior. In the second hypothesis (Kolmogorov’s first similarity hypothesis) the physical quantities defining the statistics of the small scales are defined. In particular, at high Reynolds numbers, the statistics of the small scales are universal and determined by the kinematic viscosity \( \nu \) and the dissipation rate \( \varepsilon \). Through simple dimensional analysis, the length \( \eta \), velocity \( u_\eta \) and time \( \tau_\eta \) scales of the small scales, referred to as Kolmogorov scales, can be determined as a function of \( \nu \) and \( \varepsilon \) only.

\[ \eta = \left( \frac{\nu^3}{\varepsilon} \right)^{1/4} \]  

(3.11)

\[ u_\eta = \left( \varepsilon \nu \right)^{1/4} \]  

(3.12)

\[ \tau_\eta = \left( \frac{\nu}{\varepsilon} \right)^{1/2} \]  

(3.13)

The ratio between the length of the large turbulent scales \( l_0 \) and the Kolmogorov scales \( \eta \) can be expressed as a function of the Reynolds number as \( l_0/\eta \approx Re^{3/4} \). An important consequence is highlighted: when increasing the Reynolds number, the separation between the small and large scales increases. In the third Kolmogorov hypothesis (second similarity hypothesis) Kolmogorov argued for
the existence of a range of scales between the large and the Kolmogorov scales, which are universal and dependent only on the dissipation rate $\varepsilon$. This range of scales takes the name of inertial subrange. As a consequence of the universal nature of the inertial subrange, the energy spectra for the velocity and pressure fluctuations assume the forms in Eq.3.14 and 3.15, respectively.

$$E(\kappa) = C_\kappa \varepsilon^{2/3} \kappa^{-5/3}$$  \hspace{1cm} (3.14)

$$E_{mp}(\kappa) = \alpha_p \varepsilon^{4/3} \kappa^{-7/3}$$  \hspace{1cm} (3.15)

Here, $\kappa$ is the wave number defined as $2\pi/l$.

Since turbulence is characterized by a wide range of different length and time scales, the computational resources necessary to resolve entirely the turbulent scales may be prohibitive. If we aim to resolve the turbulent scales up to the Kolmogorov scale, the spatial discretization requires a grid equal to $\Delta x \approx \eta \approx Re^{-3/4}l_0$, where $l_0$ is the characteristic length of the large scales. Similarly, the time-step must be chosen as $\Delta t \approx \Delta x/u \approx \eta/u$ since, due to accuracy and stability reasons, the fluid particle should move by one grid point at every time-step. If we combine the number of grid points and time steps as Eq.3.16 and 3.17, respectively,

$$N_x^3 \approx \left( \frac{L_{box}}{\Delta x} \right)^3 \approx \left( \frac{L_{box}}{l_0} \right)^3 Re^{9/4}$$  \hspace{1cm} (3.16)

$$N_t \approx \frac{T}{\Delta t} \approx \frac{T}{\eta/u} \approx \frac{T}{l_0/u} Re^{3/4}$$  \hspace{1cm} (3.17)

where $L_{box}$ is the characteristic size of the computational domain and $T$ the physical time, the computational time can be expressed as a cubic function of the Reynolds number as in Eq.3.18.

$$N_x^3 N_t \approx \left( \frac{T}{l_0/u} \right) \left( \frac{L_{box}}{l_0} \right)^3 Re^3$$  \hspace{1cm} (3.18)

From Eq.3.18, if we assume a computational time of one day for a case characterized by $Re = 10^3$, the computation for the same case at $Re = 10^4$ would require months. Due to the prohibitive cost of resolving all the turbulent scales for practical problems characterized by high Reynolds numbers, different models have been introduced through the years to reduce the computational cost of the simulations.

### 3.3. Reynolds-averaged Navier-Stokes equations

In RANS models the instantaneous flow variable $\phi$ is split into its mean and fluctuating component using the so-called Reynolds decomposition given in Eq.3.19.
\[ \phi = \bar{\phi} + \phi' \quad (3.19) \]

Here, the overbar denotes the Reynolds average (time or phase average) while the apex the fluctuating (zero-mean) turbulent component. In compressible flows, turbulent fluctuations cause random variations of the density. As a consequence, instead of the Reynolds decomposition, the Favre (or density-weighted) averaging \( \tilde{\phi} = \bar{\rho} \phi / \bar{\rho} \) is introduced to remove all the extra terms resulting from the product between density fluctuations and other fluctuating quantities \( (\bar{\rho}\phi'' = 0) \) (Hirsch 2007). By substituting the Favre decomposition (see Eq.3.20)

\[ \bar{\phi} = \tilde{\phi} + \phi'' \quad (3.20) \]

into the governing equations, the compressible Favre-averaged equations can be derived and read as Eq.3.21-3.23.

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_i)}{\partial x_i} = 0 \quad (3.21)
\]

\[
\frac{\partial (\bar{\rho} \tilde{u}_i)}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_i \tilde{u}_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tilde{\sigma}_{ij}}{\partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (3.22)
\]

\[
\frac{\partial (\bar{\rho} \tilde{E})}{\partial t} + \frac{\partial (\tilde{u}_j \bar{\rho} \tilde{E})}{\partial x_j} = -\frac{\partial \tilde{u}_i \bar{p}}{\partial x_j} + \frac{\partial (\sigma_{ij} \tilde{u}_i + \sigma_{ij} u''_i)}{\partial x_j}
\]

\[- \frac{\partial (\tilde{q}_j + c_p \rho u''_j T'' - \tilde{u}_i \tau_{ij} + 1/2 \rho u''_i u''_j)}{\partial x_j} \quad (3.23)\]

As a result of the averaging procedure, the Reynolds stress tensor \( \tau_{ij} = -\rho u''_i u''_j \) arises and needs to be modeled based on the mean flow variables in order to achieve problem closure. Different modeling techniques exist from simple algebraic equations, models based on a varying number of transport equations of turbulent flow variables, to models involving transport equations for the Reynolds stress tensor components. In industrial applications, the majority of the closure techniques are based on the eddy-viscosity assumption (Boussinesq 1897). In Eq.3.24, the extra contribution to the momentum diffusion given by the Reynolds stress tensor is modeled as an increase of the viscosity of the fluid through the turbulent viscosity \( \mu_t \).

\[- \rho u''_i u''_j = 2\mu_t \left( \tilde{S}_{ij} - \frac{1}{3} \tilde{S}_{kk} \delta_{ij} \right) - \frac{2}{3} \bar{\rho} k \delta_{ij} \quad (3.24)\]

As a consequence of Eq.3.24, the turbulent viscosity is a local property of the flow rather than the fluid. The Boussinesq hypothesis assumes that the transfer of momentum due to the turbulent eddies, which is represented by the Reynolds stress tensor, is proportional and aligned to the mean strain rate tensor. This
3. Governing Equations

consideration is valid in simple flows such as boundary layer and wakes while it
does not hold in complex flows.

3.3.1. Turbulence modeling

At this point, a specification for the turbulent eddy viscosity based on suitable
turbulent quantities, which are calculated through their respective transport
equations, must be provided. In the present thesis, the $k - \omega$ Shear Stress
Transport (SST) turbulence model by Menter (1992) is chosen. This closure
model has been validated for a wide variety of industrial applications (Menter
1994) and adopts the advantageous features of the $k - \omega$ model by Wilcox
(1988) in the near wall region while the $k - \varepsilon$ model by Launder & Spalding
(1983) is used outside the boundary layer. The original form of the transport
equations for the turbulent kinetic energy and specific dissipation rate by Wilcox
(1988) is given in Eq.3.25 and 3.26, respectively.

$$\frac{\partial \rho k}{\partial t} + \frac{\partial \rho u_j k}{\partial x_j} = \tau_{ij} \frac{\partial u_i}{\partial u_j} - \beta^* \rho \omega k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right]$$ (3.25)

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial \rho u_j \omega}{\partial x_j} = \frac{\gamma}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial u_j} - \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right]$$ (3.26)

Here, the turbulent dynamic viscosity is defined as $\mu_t = \rho k / \omega$ and $\tau_{ij}$ assumes
the same definition as Eq.3.24. Menter (1994) reformulates the $\omega$-equation with
the introduction of the blending function $F_1$, which is designed to be equal
to one in the boundary layer, thus restoring the original equation for $\omega$, and
progressively switches to zero outside the boundary layer, thus returning the
transport equation for the turbulent dissipation rate (see Eq.3.27).

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial \rho u_j \omega}{\partial x_j} = \frac{\gamma}{\nu_t} \tau_{ij} \frac{\partial u_i}{\partial u_j} - \beta^* \rho \omega^2 + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] +$$

$$+ 2(1 - F_1) \frac{\rho \sigma_\omega \omega^2}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}$$ (3.27)

Lastly, a modification in the definition of the turbulent viscosity is applied in
contrast to the original one given by Wilcox (1988) as in Eq.3.28.

$$\mu_t = \frac{\rho a_1 k}{\max(a_1 \omega, \Omega F_2)}$$ (3.28)

Here, $a_1$ is a model constant, $\Omega = (\partial u / \partial y)$, and $F_2$ is a blending function that
is one in the boundary layer and zero in the free shear layer. The modifications
proposed by Menter (1994) have the effect of improving the behavior of the
turbulence model in adverse pressure gradients and separating flows, making it
a suitable choice among the different two-equation turbulence models in the
context of turbocharger applications.
3.4. Large eddy simulations

LES models represent a valid alternative as compared to direct numerical simulations (DNS). The main idea at their basis relies on the explicit resolution of the large turbulent scales, which are anisotropic and dependent on the geometry, while the small ones, which exhibit a universal behavior in a statistical sense, are modeled by a sub-grid scale (SGS) model. This is achieved by applying a spatial filter to the Navier-Stokes equations, which essentially acts as a low-pass filter allowing the resolution of the large turbulent scales. The filtering can be obtained as a convolution of a function with a filtering kernel as in Eq.3.29.

$$\bar{\phi}(x_i) = \int_V G(x_i, x'_i) \phi(x'_i) \, dx'_i$$  \hspace{0.5cm} (3.29)

Here, $\bar{\phi}$ represents the spatially-filtered version of $\phi$ while $G$ is the filtering kernel. As a result, the velocity field is decomposed into a resolved and sub-grid scale component. Practically, in most CFD solvers no explicit filtering is applied but it is the grid itself acting as a filter. The filtered governing equations can be obtained by substituting Eq.3.29 into the governing equations and filtering again. Despite the different nature, the result of such operation is similar to the RANS equations reported in Section 3.3. The appearance of a sub-grid stress tensor $\tau_{ij}$ requires the introduction of a model analogous to the Reynolds stress tensor. The approach followed to model such terms makes use, similarly to RANS, of the turbulent viscosity hypothesis so that the deviatoric part of the sub-grid stress tensor is modeled as Eq.3.30.

$$\tau_{ij}^d = \tau_{ij} - \frac{\tau_{kk}}{3} \delta_{ij} = 2\bar{\nu}_{sgs} \left( \tilde{S}_{ij} - \frac{1}{3} \tilde{S}_{kk} \delta_{ij} \right)$$  \hspace{0.5cm} (3.30)

3.4.1. Sub-grid scale modeling

The simplest SGS model is the one proposed by Smagorinsky (1963), who sets the turbulent viscosity based on the cell characteristic length ($\Delta = V^{1/3}$) and the resolved strain tensor ($\tilde{S}_{ij}$) as in Eq.3.31.

$$\nu_{sgs} = (C_s \Delta)^2 |\tilde{S}|$$  \hspace{0.5cm} (3.31)

Here, $C_s = 0.1 - 0.2$ represents the Smagorinsky constant while $|\tilde{S}|$, the average strain rate based on the resolved flow, is defined as $|\tilde{S}| = (2\tilde{S}_{ij}\tilde{S}_{ij})^{1/2}$. In Eq.3.31 a substantial difference is highlighted between the SGS and the RANS turbulence model. Indeed, while the definition of turbulent viscosity for RANS models in Eq.3.28 is grid independent, the SGS model exhibits a dependence on the cell size. As a consequence, LES becomes mesh independent to the limit of a DNS, a condition where all the turbulent scales are resolved.

The Smagorinsky model suffers from high damping of the turbulent fluctuations close to the wall leading to an incorrect prediction of the skin friction.
coefficient. To alleviate this problem, Nicoud & Ducros (1999) propose a Wall-Adapting Local Eddy-Viscosity (WALE) model that forces the SGS viscosity to go to zero at the wall without the need for damping functions or special treatments for $C_s$. According to Nicoud & Ducros (1999), the SGS viscosity is reformulated as in Eq.3.32.

\[ \nu_{\text{sgs}} = (C_w \Delta)^2 \frac{(S_{ij}^d S_{ij}^d)^{3/2}}{\left(\tilde{S}_{ij} \tilde{S}_{ij}\right)^{5/2} + (S_{ij}^d S_{ij}^d)^{5/4}} \]  

(3.32)

Here, the SGS viscosity is not only a function of the resolved strain rate tensor ($\tilde{S}_{ij}$) but also of the rotating rate tensor $S_{ij}^d$ (see Eq.3.33).

\[ S_{ij}^d = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} \frac{\partial \tilde{u}_l}{\partial x_l} + \frac{\partial \tilde{u}_j}{\partial x_l} \frac{\partial \tilde{u}_l}{\partial x_i} \right) - \frac{1}{3} \delta_{ij} \frac{\partial \tilde{u}_k}{\partial x_l} \frac{\partial \tilde{u}_i}{\partial x_k} \]  

(3.33)

The LES with the WALE sub-grid scale model is adopted in Paper 1 and Paper 2.

### 3.5. Boundary layer theory

In turbocharger turbine applications, the high velocities of the flow and the high temperatures of the gases generate strong viscous and thermal gradients in the wall-near region where the flow must adhere to the no-slip condition. Both viscous ($\delta$) and thermal ($\delta_T$) boundary layer thicknesses are defined as the distance from the wall to the height where the velocity and temperature are 99% of the core velocity and temperature, respectively. The ratio between the viscous and thermal boundary layer thicknesses can be expressed as a function of the Prandtl number, which is defined in Eq.3.34.

\[ Pr = \frac{\nu}{\alpha} = \frac{c_p \mu}{k} \]  

(3.34)

Here, $\nu$ and $\alpha$ represent the kinematic viscosity of the fluid and the thermal diffusivity ($\alpha = k/(c_p \rho)$). The Prandtl number expresses the ratio between the momentum and thermal diffusivity. When the Prandtl number is approximately unity, viscous and thermal boundary layers are characterized by the same thickness. When $Pr \gg 1$, the diffusion of momentum dominates and the viscous boundary layer thickness is larger than the thermal boundary layer one. When $Pr \ll 1$, thermal diffusivity dominates and the viscous boundary layer thickness is smaller than the thermal boundary layer one. If air is considered as a working fluid (as done in this thesis) the Prandtl number assumes the value of 0.71 for a large range of temperatures so that the viscous boundary layer is thinner than the thermal boundary layer.

### 3.6. CFD for rotating machinery

Numerical simulations play an important role in turbomachinery design and development. Indeed, they enable the evaluation of different designs before
more expensive and time-consuming experimental campaigns. For rotating machines in CFD applications, the computational domain requires to be split in two different regions: stationary and rotating. Nowadays, different models to reproduce the rotating motion are available and, at the highest level, they can be divided into steady-state and unsteady methodologies.

In steady-state approaches, the rotating motion is modeled by using the Moving Reference Frame (MRF) method. The governing equations in the rotating region are solved in the reference frame of rotation, while the flow variables are interpolated between the stationary and the rotating regions across the stator-rotor interface. Two different approaches are available for the stator-rotor interface treatment: the frozen-rotor and mixing-plane. While the relative position of the rotor to the stator is maintained fixed, the two approaches differ in the method used to interpolate the flow variables across the stator-rotor interface. In the frozen-rotor approach, the primitive flow variables are simply interpolated from one reference system to the other. The advantages of the frozen-rotor approach are represented by its simplicity and low computational cost. However, since the position of the rotor is maintained fixed, the results are position-dependent, characteristic also known as clocking effect. To overcome the limitation of the frozen-rotor approach, Denton & Singh (1979) introduced the concept of mixing-plane to represent an average solution of the full rotation of the rotor. Both in its explicit or implicit formulation (and despite the different implementations) the mixing-plane is based on circumferential averaging of the flow at the stator-rotor interface. The circumferential average of the flow variables makes the results independent of the stator-rotor relative position. However, since the flow is abruptly mixed at the interface, a higher entropy rise is induced (Fritsch & Giles 1995). Frozen-rotor and mixing-plane approaches have gained popularity in industrial practices due to their low computational cost. Specifically, mixing-plane is largely used to simulate single-blade passages of the rotor, thus reducing the size of the computational domain. However, both methods can only describe the turbomachines’ behavior in terms of averaged performance. As opposed to steady-state approaches, unsteady methods are based on the Sliding Mesh Method (SMM). At every time-step, the rotating region is physically rotated and the flow variables are interpolated across the stator-rotor interface. Due to the need of recomputing the position of the rotor at each physical time-step, the Sliding Mesh Method is computationally more expensive than the frozen-rotor and mixing-plane. However, this approach enables a dynamic and more accurate description of the stator-rotor interaction.

3.7. Numerical methods

The governing equations and their constitutive relations represent a system of non-linear partial differential equations that admit an analytical solution only for simplified cases. For all other cases of practical interest, numerical methods are adopted and are based on the spatial and temporal discretization of the domain of interest together with the governing equations. Many open-source and commercial solvers adopt the Finite Volume Method (FVM) for the
discretization of the governing equations, which can be expressed in terms of an integral transport equation for a generic conservative variable \( \phi \) over a volume \( V \) as in Eq. 3.35.

\[
\frac{d}{dt} \int_V \rho \phi \, dV + \int_A \rho \phi (\mathbf{v} \cdot \mathbf{n}) \, dA = \int_A \Gamma_\phi (\nabla \phi \cdot \mathbf{n}) \, dA + \int_V S_\phi \, dV \tag{3.35}
\]

Here, \( \Gamma_\phi \) represents the diffusion coefficient of the general quantity \( \phi \), \( V \) the volume of interest, and \( \mathbf{v} \) the velocity vector in the absolute reference system. The transport equation must be fulfilled in each cell of the domain in its discrete form as in Eq. 3.36, where the summation symbol is meant all over the faces of the cell considered.

\[
\frac{d}{dt} (\rho \phi V)_0 + \sum_f [\rho \phi (\mathbf{v} \cdot \mathbf{n}) a]_f = \sum_f [\Gamma_\phi (\nabla \phi \cdot \mathbf{n}) a]_f + (S_\phi V)_0 \tag{3.36}
\]

For unsteady problems, time is an additional coordinate that needs to be discretized as well as spatial coordinates. The solution at a specific time is obtained from a certain number of previous time-levels that determines the order of the numerical scheme. In this thesis, the transient term is discretized by the second-order backward difference scheme defined in Eq. 3.37.

\[
\frac{\partial}{\partial t} (\rho \phi V) \approx \frac{3(\rho \phi V)_{n+1} - 4(\rho \phi V)_n + (\rho \phi V)_{n-1}}{2\Delta t} \tag{3.37}
\]

Here, \( \Delta t \) represents the time-step while the subscripts \( n + 1 \), \( n \), and \( n - 1 \) refer to the solution at the current and previous two time-levels.

The surface fluxes require the definition of the necessary quantities at the cell faces based on the ones defined at the cell centers. The simplest approximation is represented by the First Order Upwind (FOU) scheme as in Eq. 3.38. In this case, the value at the face \( f \) is set equal to the value at the cell center depending on the direction of the mass flux.

\[
(\dot{m}_f \phi)_f = \begin{cases} 
\dot{m}_f \phi_0 & \dot{m}_f \geq 0 \\
\dot{m}_f \phi_1 & \dot{m}_f < 0 
\end{cases} \tag{3.38}
\]

Here, the subscripts 0 and 1 identify two different neighboring cells sharing the same face. It can be demonstrated that the nature of the approximation error of this scheme is of diffusive type (LeVeque et al. 2002). This characteristic contributes to stabilizing the numerical solution. However, since the face value is approximated with the cell center one, this numerical scheme is characterized by a low accuracy when the face normal is not aligned with the flow direction. To increase the accuracy and limit the numerical diffusion, the Second Order Upwind (SOU) scheme is preferred since the value at the face is linearly interpolated from the values at the center of the cells.
Another scheme that exhibits interesting properties is the central differencing scheme (CDS) which approximates the value at the face center with a linear interpolation between two cells center values (see Eq.3.40).

\[
(\dot{m}_f \phi)_f = \begin{cases} 
\dot{m}_f \phi_{f,0} = \dot{m}_f [\phi_0 + (x_f - x_0) \cdot (\nabla \phi)_0] & \dot{m}_f \geq 0 \\
\dot{m}_f \phi_{f,1} = \dot{m}_f [\phi_1 + (x_f - x_1) \cdot (\nabla \phi)_1] & \dot{m}_f < 0
\end{cases}
\] (3.39)

Here, \( f \) is a linear interpolating factor. Differently from the upwind schemes, the error of the CDS is of dispersive nature and may lead to unphysical oscillations. However, it has the advantage of preserving the turbulent kinetic energy making it suitable for LES. A good compromise between the above mentioned schemes is the hybrid second order upwind/central-differencing scheme (see Eq.3.41). It guarantees the stability of the upwind scheme together with the accuracy of the central-differencing one.

\[
(\dot{m}_f \phi)_f = \begin{cases} 
\dot{m}_f \{\sigma \phi_{f,0} + (1 - \sigma)[f \phi_{f,0} + (1 - f)\phi_{f,1}]\} & \dot{m}_f \geq 0 \\
\dot{m}_f \{\sigma \phi_{f,1} + (1 - \sigma)[f \phi_{f,0} + (1 - f)\phi_{f,1}]\} & \dot{m}_f < 0
\end{cases}
\] (3.41)

Here, \( \sigma \) represents a blending function that restores the SOU scheme for a value equal to 1 and the CDS scheme for a value equal to 0. For the LES simulations, this scheme is chosen due to its favorable properties.
Chapter 4

Summary of results

In this chapter, a summary of the main results associated with papers embedded in the second part of the thesis is presented. At first, comparisons between CFD data and available experimental measurements in terms of performance parameters are carried out for validation purposes in Section 4.1. Meshing considerations for the specific model adopted are discussed and the different solutions are evaluated. Then, the results are divided into two categories. In Section 4.2, the main characteristics of the flow inside the hot-side system are investigated. The results are part of Paper 1 and Paper 2, where Large Eddy Simulations have been used to investigate the complex flow phenomena developing in both continuous and pulsating flow conditions. In Section 4.3, which includes results detailed in Paper 3-7, the focus is primarily oriented to the analysis of the unsteady performance of the hot-side system and the development of reduced-order models for the prediction of the turbine unsteady performance. The studies presented in the second part make use of Reynolds-averaged Navier-Stokes models and Detached Eddy Simulations.

4.1. Model validation

As detailed in Chapter 2, turbocharger radial turbines operate in two distinct conditions: continuous and pulsating flow conditions. At first, the computational model is validated against experimental results obtained in hot gas-stand continuous flow conditions and provided by the manufacturer. This represents a common practice for turbocharger radial turbine numerical models validation (Galindo et al. 2013b). For the validation, system A (see Fig.2.3) is considered and a steady-state RANS model is adopted to cover the entire performance map in a range of rotational speeds between 48479 rpm and 149386 rpm. Such a wide range of rotational speeds allows us to assess the generality of the model to a large range of operating conditions. The computational cost for covering the performance map is reduced by the use of a steady-state RANS formulation coupled with a moving reference frame (MRF) method such that, in the rotating part, the governing equations are solved in the rotating reference frame. The stator-rotor interface is treated by an implicit mixing-plane approach to avoid the clocking effect, i.e. the effect associated with the relative position between the volute and rotor.
4.1. Model validation

In Fig.4.1, the performance map is presented as mass flow parameter (MFP in Eq.2.3) against the expansion ratio (\(\pi\) in Eq.2.4). The inlet physical quantities concurring in the definition of MFP and \(\pi\) are calculated by mass-flow averaging at the inlet pipe-volute section (\(S_1\) with reference to Fig.2.3) while the outlet pressure is measured at the outlet section (\(S_4\)) and is imposed equal to atmospheric conditions (\(p_{out} = 101.3\, \text{kPa}\)). It can be noticed that the numerical results approximate accurately the experimental data with a maximum percentage deviation equal to 3%. In Fig.4.1b, the LES model validation (from Paper 1) is reported for two different operating points at a turbine speed equal to 72917rpm. The deviations are reported smaller than 3% for both the peak efficiency point and the highest expansion ratio point in the performance line. Given the small errors reported, which may be caused by unavoidable differences in the geometry and uncertainties in the boundary conditions, both models can be considered validated.

4.1.1. Mesh considerations

In all cases analyzed, the computational domain is discretized by polyhedral elements. This choice has two advantages. Firstly, it allows for automatic mesh generation, which is preferable when dealing with highly-complex geometries as the one considered in the present thesis. Secondarily, polyhedral elements are characterized by higher connectivity to the neighboring cells than tetrahedral and hexahedral elements, leading to a reduction of the numerical diffusion (Chow et al. 1996). Differently, in the outlet convergent duct, where the flow is expected to be more aligned with the center line, a hexahedral mesh is generated by extrusion of the polyhedral grid.
The flow field characteristics significantly vary in the different components of the system. As a consequence, a non-uniform mesh density is adopted. The rotating region, since it is characterized by larger velocity gradients compared to the other parts of the system, is discretized with a mesh density 4 times larger than the volute region (with reference to the mesh used in Paper 6 and Paper 7). The resolution in the near-wall region is subject to a sensitivity study since heat transfer and turbine torque require a careful evaluation of the resolution of the flow in the near wall-region. To assess the correct resolution of the viscous and thermal boundary layer, a mesh sensitivity study is performed by considering different wall resolutions. Instead of considering a continuous flow scenario, the sensitivity of the turbine torque and heat transfer rate with respect to the near-wall resolution are investigated in pulsating flow conditions. In Fig.4.2a and 4.2b, the instantaneous turbine torque and heat transfer rate at the exhaust manifold are evaluated for increasing near wall-resolution (from G1 to G4). It can be seen how, while the turbine torque is almost insensitive in the range of wall resolutions considered, the heat transfer rate requires a larger wall resolution than the turbine torque. The percentage difference between grid G1 (coarse) and grid G4 (fine) is reported equal to 10%, while the heat transfer rate becomes mesh independent starting from G3. As a consequence, the mesh resolution used in G2 is adopted at the rotor surfaces, while the mesh resolution used in G3 on the other components surfaces. As a result of the wall resolution adopted, the non-dimensional wall distance $y_+$ is kept below unity for the entire pulse period.

The above-mentioned considerations on the mesh resolution are commonly adopted for both RANS and LES models. However, the two methods significantly differ in the nature of the turbulence modeling. Indeed, since in a RANS model all the turbulent scales are modeled, the turbulence model is independent of the
grid resolution. On contrary, the sub-grid scale model in an LES is affected by the cell characteristic length, which determines the spatial filter between resolved and unresolved turbulent scales (see Eq.3.31 and 3.32). As a consequence, an LES becomes mesh independent when all the turbulent scales are resolved and a DNS resolution is reached. Assessing the grid resolution for LES applications is an important step. Indeed, since the cell size determines the spatial filter, thus the threshold between resolved and unresolved turbulent scales, a preliminary investigation of the mesh characteristics is necessary to guarantee the resolution of the turbulent scales in the inertial sub-range. In the literature, many different criteria and indices are proposed to determine the appropriate grid resolution for LES approaches. The interested reader is referred to the comprehensive overview given by Celik et al. (2005); di Mare et al. (2014) while only the most common approaches are discussed in the following. Since only a fraction of the total turbulent kinetic energy is resolved in an LES, the energy ratio in Eq.4.1, defined as the ratio between the resolved and total turbulent kinetic energy, is often used to assess the quality of the grid resolution.

\[
ER = \frac{k_{res}}{k_{res} + k_{sgs}}
\]

Here, \(k_{res}\) represents the part of turbulent kinetic energy resolved by the filtered-equations, while \(k_{sgs}\) is the part modeled by the sub-grid scale model. According to Pope (2004), in a well-resolved LES, the energy ratio should exceed the 80%. A second approach to assess the quality of the grid is given by local spectra analysis. Since the turbulent kinetic energy and pressure fluctuations spectra are described by universal laws in the inertial sub-range, a suitable grid for LES must be able to replicate the \(-5/3\) and \(-7/3\) trends. Although this approach is effective in fundamental applications where the geometry is simple and is not constituted by regions with different geometrical characteristics, such as boundary layer, channel, and pipe flows, it is not suitable for complex geometries, such as the one considered here, where the mesh characteristics largely vary in the different components of the system. Consequently, a local estimator is not representative of the global mesh characteristics.

Since a well-resolved LES requires resolving the turbulent scales in the inertial sub-range, the ratio between the Taylor micro-scale \(\lambda\) and the cell characteristic length \(\Delta = V^{1/3}\) is chosen as a quality index in the present work. The Taylor micro-scale is here calculated as Eq.4.2,

\[
\lambda = \sqrt{\frac{10\nu k}{\varepsilon}}
\]

where \(\nu\) represents the kinematic viscosity, \(k\) is the turbulent kinetic energy, and \(\varepsilon\) is the turbulent dissipation rate. The cell characteristic length is otherwise calculated as the cube root of the local cell volume. In our applications, we target a value of the Taylor-ratio equal to unity in order to guarantee the resolution of the turbulent scales up to the Taylor micro-scale. This condition is
4. Summary of results

Figure 4.3: Comparison of the cell characteristic length ($\Delta = V^{1/3}$) for the initial mesh (left) and refined mesh (right) at the volute and rotor sections.

Figure 4.4: Comparison of the cell characteristic length ($\Delta = V^{1/3}$) for the initial mesh (left) and refined mesh (right) for the blade passage.

achieved in two steps. At first, a steady-state RANS simulation is run in order to calculate the Taylor micro-scale field. Then, the computational domain is re-meshed targeting a cell characteristic length equal to the Taylor micro-scale. Such methodology resembles the method used by Pietroniro et al. (2022) on the compressor side. Following this approach, a uniform resolution is reached in the entire domain in terms of turbulent scales resolved.

Examples of mesh refinement for continuous flow conditions are given in Fig.4.3, for the mesh refinement of the full-geometry in Paper 1, and in Fig.4.4 for the mesh refinement of the blade passage set-up used in Paper 2. In both cases, it can be noticed that the initial mesh is characterized by a uniform resolution in regions that display similar geometrical characteristics. For example, in Fig.4.3, the mesh resolution in the rotor is uniform except for the near-wall region while, in Fig.4.4, the cell characteristic length increases
uniformly with the distance from the wall. After the mesh refinement, the cell resolution does not depend on the geometrical characteristics but on the local state of the flow. For example, in both cases, a higher mesh resolution is achieved on the blade suction side and in the wake region, where significant turbulent activities occur.

4.2. Flow analysis

The hot-side system is constituted by components of different sizes and shapes which represent a fundamental case on their own. As a consequence, it is primarily important to analyze the main flow characteristics of the singular components. In Section 4.2.1, we focus on the investigation of the volute-rotor interaction in continuous flow conditions while, in Section 4.2.2, the aerodynamic performances of the turbine are investigated in both continuous and pulsating flow conditions.

4.2.1. Volute-rotor interaction

The hot-side system is constituted by complex components with different geometrical characteristics that interact with each other. The volute has the role of redirecting the flow perpendicularly to the axis of rotation of the turbine and uniformly in the circumferential direction. However, non-uniform inflow conditions are caused by the presence of the volute tongue, where a wake is generated from the flow separation on its surface. According to Pan et al. (2022), the unsteady forces generated by the non-uniformity in the inflow conditions at the volute-rotor interface may increase the risk of high cycle fatigue (HCF) failure. In Paper 1, the volute-rotor interaction is investigated using Large Eddy Simulations. In Fig.4.5a, the phase-averaged blade torque is plotted

Figure 4.5: Phase-averaged blade torque against the circumferential position (Fig.4.5a) and time-average and standard deviation of the relative inflow angle (Fig.4.5b) against the circumferential positions.
4. Summary of results

θ = 0°  θ = +10°
θ = +20°  θ = +30°
θ = +40°  θ = +50°
Low  High

Figure 4.6: Snapshots of the specific entropy field at the volute and rotor mid-span sections. The colorbar is saturated so that blue and red areas correspond to low and high values of the specific entropy.

against the circumferential position. The origin of the reference system is set as the tangent at the volute tongue from the center of rotation. The blade torque exhibits significant deviations in the circumferential direction, with a maximum 25% larger than the time-averaged turbine torque at the circumferential position θ = 40°. The causes of this trend can be related to the non-uniform aerodynamic inflow conditions. In Fig.4.5b, the time-averaged value and standard
Figure 4.7: Snapshots of the specific entropy field at the rotor-mid span. The colorbar is saturated so that blue and red areas correspond to low and high values of the specific entropy. The blade subject of the analysis is highlighted by a red dot.

deviation of the relative inflow angle are plotted against the circumferential position. It can be seen that the relative inflow angle assumes negative values at the circumferential position $\theta = 0^\circ - 40^\circ$. Here, the flow is characterized by a large radial velocity component that is induced by the flow acceleration at the volute tongue. Moreover, due to the flow separation developing at the volute tongue, the tangential component of the velocity is smaller as compared to other circumferential positions. As a consequence, the flow is better aligned to the turbine blade and the blade performance improves. In Fig.4.6 and 4.7, the flow dynamics of the volute-rotor interaction is investigated by flow visualizations of the specific entropy at the volute and rotor mid-span sections. The blade analyzed in the following is highlighted by a red dot in Fig.4.7. At position $\theta = 0^\circ$, the turbine blade is not affected by the tongue wake. As the rotation continues, the turbine blade starts interacting with the tongue wake at a position $\theta = +10^\circ$. Subsequently, the tongue wake is chopped by the blade leading edge
and a small vortical structure develops at the blade leading edge on the pressure side ($\theta = +20^\circ$). The tongue wake on the suction side convects downstream while it is stretched due to the main flow acceleration ($\theta = +30^\circ$ and $\theta = +40^\circ$). Lastly, at position $\theta = +50^\circ$, the flow on both pressure and suction sides is characterized by low entropy values since the flow is better aligned than the other circumferential positions.

Based on the results discussed, the non-uniformity in the inflow conditions has a significant impact on the blade performance. It is therefore interesting to compare the results observed with a steady RANS model (with a mixing-plane approach for the treatment of the stator-rotor interface) to quantify the effects of neglecting the interaction between the tongue wake and the blade. In Fig.4.8, the instantaneous and time-averaged blade torque predicted by the LES model and the averaged blade torque predicted by the steady RANS model are compared. The percentage difference in the averaged torque between the LES and steady RANS models is reported equal to 0.6%. As a consequence, the computational cost of predicting the averaged turbine performance can be reduced by using steady RANS models.

4.2.2. Rotor flow analysis

During pulsating flow conditions, the turbine blade is subject to a wide range of operating conditions both in terms of magnitude and direction of the velocity field. As a consequence, the turbine performance largely vary in time and different flow structures and secondary flows develop. Examples of the turbine performance in both continuous and pulsating flow conditions are reported in Fig.4.9, where time-varying turbine torque and isentropic efficiency are reported. It can be seen that the instantaneous turbine torque assumes negative values at the beginning of the pulse when low mass flow rates occur. In this case, the rotor is at a state called free-wheeling condition, and the transfer of momentum
is directed from the rotor to the flow. As the mass flow rate increases, the torque increases as well reaching a peak that occurs after the peak of the mass flow rate pulse. In terms of averaged performance, the turbine torque measured in pulsating flow conditions increases by 24% as compared to corresponding continuous flow conditions. In Fig. 4.9b, the instantaneous isentropic efficiency experiences a global minimum at negative values of the turbine torque and a local minimum close to the turbine torque peak. In terms of averaged performance, the isentropic efficiency is characterized by an opposite trend with respect to the turbine torque since it reduces by 17% from continuous to pulsating flow conditions. As a consequence, under pulsating inflow conditions, the expansion inside the blade passage departs more from an ideal isentropic process due to the wide range of relative inflow angles that the turbine is exposed to. Indeed, during pulsating flow conditions, the relative inflow angle varies between a minimum of $-80^\circ$ to a maximum of $+39^\circ$. These two extreme inflow conditions display characteristic secondary flows that develop in the blade passage. In Fig. 4.10 and 4.11, the instantaneous relative Mach number and specific entropy are reported for relative inflow angles equal to $-60^\circ$ and $+20^\circ$, respectively, corresponding to low and high mass flow rate conditions. At low mass flow rates, the flow field on the pressure side is dominated by a flow recirculation, whose diameter is of the same order as the blade passage, that develops from the flow separation at the blade leading edge. In Fig. 4.12, the $\lambda_2$ criterion highlights the vortical structure originating from the flow separation on the blade pressure side and convecting downstream in the proximity of the blade tip. At high mass flow rates, the relative inflow angle grows to high positive values so that the flow separates at the blade suction side (Fig. 4.11). A richer

![Figure 4.9: Time-varying turbine torque (Fig.4.9a) and isentropic efficiency (Fig.4.9b) over the pulse cycle for pulsating and continuous flow conditions. Subscripts $p$ and $s$ denotes pulsating and continuous flow conditions while $a$ denotes the averaged performance of the turbine in pulsating flow. Plots are adapted from Paper 2.](image-url)
variety of flow structures can be observed by the \( \lambda_2 \) criterion in Fig.4.13. The main vortical structures develop at the blade leading edge as the result of the flow separation. Then, two other vortical structures can be identified. The first one originates in the inter-space between the turbine and the back-plate. The second one develops within the tip-gap between the blade and the shroud wall giving origin to the so-called tip-leakage vortex, which is the result of the pressure difference between the pressure and suction sides.

This analysis highlights that the turbine, during pulsating conditions, operates far from ideal isentropic conditions due to the flow irreversibilities that develop because of flow separations on the pressure and suction sides. As a consequence, the time-averaged isentropic efficiency in pulsating flow conditions
is lower than the one measured in corresponding continuous flow conditions. However, greater power output is reported in pulsating than in continuous flow conditions. This trend of turbine performance makes the utilization of the isentropic efficiency as an index to evaluate the performance of radial turbines during pulsating flow conditions questionable.

4.3. Performance analysis

In this section, the main topic is centered on the hot-side system response to different pulsating flow conditions. This analysis is meant to highlight
specific trends of the turbine performance with respect to variations of the pulse characteristics. In Section 4.3.1, which summarizes some of the results of Paper 3, a classic parametric study is carried out to investigate the turbine performance subject to pulses characterized by different amplitudes and frequencies. The effects of the choice of the type of boundary conditions in the context of numerical simulations are discussed in Section 4.3.2. In Section 4.3.3, the turbine performance are analyzed in terms of deviations from quasi-steady conditions. Moreover, an algebraic model is developed to predict such deviations. In Section 4.3.4, a reduced-order model based on a fully-connected neural network, developed and presented in Paper 4, is used to model the unsteady turbine performance in pulsating flow conditions. In Sec.4.3.5, a gradient-based algorithm developed in Paper 5 is used to determine the optimum pulse shape for the maximization of the turbine torque. Lastly, Section 4.3.6 summarizes the results of Paper 6 and Paper 7, where an exergy-based analysis is carried out to investigate the hot-side system response to pulse characteristics variations under on-engine conditions, i.e. considering heat transfer and exhaust manifold effects.

4.3.1. Parametric study

In this section, we analyze the turbine system response with respect to pulse characteristics variations. Mass flow rate and total temperature are imposed as inlet boundary conditions and are modified to vary the pulse amplitude and frequency. For the analysis of the pulse amplitude effects, the mass flow rate and total temperature pulses are modified according to Eq.4.3 used by Zhao et al. (2018).

$$\Psi = \Psi_{ave} + \phi (\Psi_{base} - \Psi_{ave})$$  \hspace{1cm} (4.3)

Here, $\Psi$ is a generic variable (mass flow rate or total temperature), $\phi$ is the normalized pulse amplitude, and the subscripts $base$ and $ave$ refer to the baseline pulse and its mean value. This approach is considered appropriate for this type of studies since the amplitude is varied while maintaining the same mean value over the pulse. For the pulse amplitude analysis, the normalized amplitude is chosen equal to $\phi = 0.4, 0.7, 1.0, 1.3, 1.6$ (at $f = 80$ Hz). For the frequency analysis, the pulse frequency is set equal to $f = 20, 40, 60, 80, 100$ Hz (at $\phi = 1.0$). The boundary conditions are reported in Fig.4.14 for the amplitude and frequency analysis, respectively.

In Fig.4.15a and 4.15b, the time-averaged expansion ratio and isentropic efficiency (see Eq.2.7) are reported with respect to variations of the pulse amplitude and frequency. Both quantities experience characteristic trends. With increasing pulse amplitude, the time-averaged expansion ratio increases by 4.0% while the isentropic efficiency decreases by 6.5% from case $\phi = 0.4$ to case $\phi = 1.6$. The rise in the expansion ratio can be attributed to the increase in the total pressure at the inlet of the system. Indeed, the time-averaged total pressure increases by 12% from case $\phi = 0.4$ to case 1.6 showing that, with increasing
pulse amplitude, a larger work is required for pumping the same amount of air in the system due to the larger resistance of the flow. With increasing pulse amplitude, an opposite trend is reported for the isentropic efficiency and can be related to the wider range of inflow conditions the turbine blade is subject to. Since a constraint on the mean value is applied when generating the boundary conditions through Eq.4.3, case $\phi = 1.6$ is characterized by a wider range of operating conditions, i.e. lower minimum and larger maximum values of the mass flow rate. As a consequence, the range of relative inflow angles is also larger, so that the turbine blade operates in off-design conditions for a larger fraction of the pulse period. In Fig.4.16a, the time-averaged relative inflow angle, together with the minimum and maximum instantaneous values registered during the pulse, is reported with respect to pulse amplitude variations. It can be noticed that the time-averaged value slightly decreases while the range significantly expands. The range of relative inflow angles is reported equal to $[-21^\circ; +31^\circ]$ and $[-53^\circ; +38^\circ]$ for cases $\phi = 0.4$ and $\phi = 1.6$, respectively.

For the pulse frequency study, increasing frequency has a similar but opposite effect as compared to the pulse amplitude. With increasing frequency, the time-averaged expansion ratio decreases by 1.3% from case $f = 20$ Hz to case $f = 100$ Hz, while the isentropic efficiency increases by 6.5%. Similarly to the amplitude analysis, the decrease in the expansion ratio can be related to the time-averaged total pressure at the inlet of the system. Indeed, increasing the pulse frequency causes the time-averaged total pressure to decrease, so lower pumping work is required. On the other side, the isentropic efficiency increases because the range of inflow conditions reduces from case $f = 20$ Hz to case $f = 100$ Hz (see Fig.4.16b). The range of relative inflow angles is reported equal to $[-65^\circ; +41^\circ]$ and $[-30^\circ; +32^\circ]$ for cases $f = 20$ Hz and $f = 100$ Hz,
4. Summary of results

![Graphs](a) (b)

Figure 4.15: Time-averaged expansion ratio and isentropic efficiency for the amplitude (Fig.4.15a) and frequency (Fig.4.15b) parametric studies.

respectively. As a consequence, at higher frequencies, the turbine blade operates in off-design conditions for a smaller fraction of the pulse period.

4.3.2. Influence of the type of BCs

In compressible flow solvers, two combinations of inlet boundary conditions are generally imposed: total conditions (total pressure and temperature) or mass flow rate and total temperature. In parametric studies involving variations of the pulse characteristics such as pulse amplitude and frequency, the choice of the combination of boundary conditions has an impact on the performance trend since it determines the type of constraint imposed. When pulse amplitude and frequency are modified as in Section 4.3.1, the time-averaged total pressure or mass flow rate is conserved among the different pulses depending on the type of boundary conditions imposed. In all works (except Paper 5) constituting this thesis, the mass flow rate is imposed instead of total pressure as an inlet
boundary condition together with the total temperature. The reasons behind this choice rely on the fact that mass is a conservative variable and, when modifying the pulse amplitude, it is possible to prevent backflow from occurring at the system inlet which may lead to numerical instabilities of the computational model. Moreover, from an engine perspective, this corresponds to assuming the same amount of air inside the cylinder for different pulse conditions.

To assess how the combination of boundary conditions affects the turbine behavior, an amplitude parametric study has been performed in Paper 5 by imposing total pressure instead of mass flow rate as an inlet boundary condition. The total pressure pulse has been modified through Eq.4.3 while the isentropic relation \( p_1^\gamma - \gamma T_1^\gamma = \text{const} \) has been used to derive the total temperature pulse as done by Galindo et al. (2013). In Tab.4.1, the effect of the choice of the inlet boundary conditions \( \dot{m} + T_i \) or \( p_t + T_i \) is qualitatively summarized by comparing different time-averaged turbine performance indicators for increasing pulse amplitude. It can be noticed that, for the pulse amplitude study, the trend of the expansion ratio and turbine torque is opposite whether \( \dot{m} + T_i \) or \( p_t + T_i \) are used as boundary conditions. These opposite trends of the turbine performance can be explained by analyzing the trend of the independent variable (IV) measured at the inlet section. For the combination \( \dot{m} + T_i \), the time-averaged total pressure increases while, for the combination \( p_t + T_i \), the time-averaged mass flow rate decreases. In the first case, as pulse amplitude increases, the flow resistance inside the domain grows and, as a consequence, a larger work is required to pump the air inside the system. This mechanism causes the time-averaged total pressure to increase contributing to the growth of the expansion ratio and turbine torque. For the combination \( p_t + T_i \), the time-averaged total pressure is locked by the constraint applied through Eq.4.3. As a consequence, with increasing pulse amplitude, less air is pumped into the

Figure 4.16: Minimum, maximum, and time-averaged relative inflow angle for the amplitude (Fig.4.16a) and frequency (Fig.4.16b) parametric studies.
Table 4.1: Qualitative variations of the turbine performance for different combinations of boundary conditions as pulse amplitude increases.

<table>
<thead>
<tr>
<th>Param BC IV</th>
<th>Performance</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\phi$</td>
<td>$\dot{m}<em>{ave}$ $p</em>{t, ave}$ $\pi$ $\eta_{ls}$ $\tau$</td>
</tr>
<tr>
<td>$\uparrow$</td>
<td>$\uparrow$ $\uparrow$ $\uparrow$</td>
</tr>
<tr>
<td>$\downarrow$</td>
<td>$\downarrow$ $\downarrow$ $\downarrow$</td>
</tr>
</tbody>
</table>

system due to the larger flow resistance. As a consequence, the expansion ratio and turbine torque decrease with increasing pulse amplitude.

The results highlight that, in the context of numerical parametric studies, constraints on the time-averaged total pressure or mass flow rate influence the turbine performance trends in the opposite way. It is therefore important to define standards for the choice of the type of inlet boundary conditions appropriate to simulate real on-engine conditions in turbocharger turbine applications. Most of the numerical works mentioned in Section 1.3 adopt the total pressure instead of the mass flow rate as an inlet boundary condition due to the connection with the definition of the expansion ratio in the performance map. However, if the expansion ratio is measured at sections far away from the rotor, e.g. inlet and outlet of the system, the index represents the performance of the entire system instead of the turbine alone. Consequently, the expansion ratio loses its original meaning and, in the context of numerical simulations, assumes the role of a boundary condition. In this thesis, it is considered more appropriate to impose the mass flow rate instead of total pressure as an inlet boundary condition in parametric studies since it corresponds to assuming the same amount of air expelled by the cylinders of the engine.

4.3.3. Hysteresis loop

This section focuses on the characterization of the performance deviations from quasi-steady to pulsating flow conditions and summarizes some of the findings of Paper 3. In Fig.4.17, the hysteresis loop as mass flow parameter against expansion ratio is reported for pulse amplitude and frequency variations, respectively. The trend of the hysteresis loop response is clear: with increasing pulse amplitude and frequency, the deviations from quasi-steady conditions increase. However, the classical representation used in Fig.4.17a and 4.17b does not provide any information about the mechanisms regulating the deviations. For this reason, the same data are reported in Fig.4.18a and 4.18b colored by the instantaneous normalized reduced frequency (proposed by Cao et al. (2014)) and reported in Eq.4.4.

$$|\varepsilon(t)|\beta(t) = \frac{|\Delta p|}{\bar{p}} \frac{t_f}{\Delta t}$$ (4.4)
4.3. Performance analysis

Figure 4.17: Hysteresis loops for the pulse amplitude (Fig.4.17a) and frequency (Fig.4.17b) parametric studies.

Here, $\Delta p$ represents the variation of the pressure at a given time-step $\Delta t$ and $p$ the average of the pressure pulse. The residence time $t_f$ is calculated by a cross-correlation between the mass flow rate pulses measured at the inlet pipe-volute interface (section $S_1$ with reference to Fig.2.3) and the rotor-diffuser interface $(S_3)$. According to Cao et al. (2014), the rotor cannot be treated as a quasi-steady device when the normalized reduced frequency exceeds a threshold equal to 0.07. In Fig.4.18a and 4.18b, the intensity of the deviations from quasi-steady performance shows a clear proportionality with the normalized reduced frequency. Indeed, the deviation is largest when $|\varepsilon(t)|\beta(t)$ is maximum, i.e. when the time derivative of the pressure pulse is largest, while the unsteady performance match the quasi-steady ones when the $|\varepsilon(t)|\beta(t)$ is null, at the minimum and maximum points of the pressure pulse. As a consequence of the results observed, it is reasonable to assume that the intensity of the deviations is not directly influenced by pulse amplitude and frequency variations, but by the slope of the pressure pulse and residence time of the fluid particle. Based on this observation, an algebraic model to predict the performance deviations from quasi-steady to pulsating flow conditions is proposed in Eq.4.5.

$$\frac{\text{MFP}_p(p(t))}{\text{MFP}_q(p(t))} = 1 + \varepsilon(t)\beta(t) \quad (4.5)$$

Here, at a given expansion ratio, the ratio between the mass flow parameter in pulsating and quasi-steady flow conditions is modeled as a linear function of the normalized reduced frequency. The model proposed in Eq.4.5 is compared to the CFD results in Fig.4.19 for case $\phi = 1.6$ and $f = 80\text{Hz}$. The comparison shows a good agreement between the algebraic model and the CFD results with an RMSE equal to 0.046.
The results show that the normalized reduced frequency is a useful non-dimensional number able to model the unsteady turbine performance under pulsating flow conditions. The process of mass accumulation inside the volute is well captured and it reflects with great accuracy in the model proposed in Eq.4.5. Furthermore, another important aspect is highlighted regarding the nature of the turbine performance deviations. It is inappropriate to correlate the size of the hysteresis loop to the pulse amplitude and frequency. Indeed, the size of the hysteresis loop is proportional to the time derivative of the pressure pulse, as shown in Fig.4.18a and 4.18b, and is indirectly influenced by variations in pulse amplitude and frequency. This consideration is used in the following section to build a data-driven model for predicting the unsteady turbine performance in pulsating flow conditions.
4.3. Performance analysis

Figure 4.19: Hysteresis loop as mass flow parameter (MFP) against expansion ratio ($\pi$) for case $\phi = 1.6$ and $f = 80$ Hz. Quasi-steady and pulsating turbine performance are reported together with the hysteresis loop predicted according to the model in Eq.4.5.

4.3.4. Performance modeling

The turbocharger-engine system represents a complex system constituted in turn by different complex sub-systems. Due to the high complexity of the problem, it is rather impractical to model the entire system through time-consuming CFD simulations. As a consequence, the most common approach is using reduced-order models to model the system’s behavior. In Paper 4, the observation that the performance deviations are proportional to the time derivative of the pressure pulse is used to develop a data-driven method to predict the instantaneous turbine torque and circumferentially-averaged relative inflow angle in pulsating flow conditions. The model is based on a fully-connected neural network that acts as a black box taking and returning certain input and output variables of interest, respectively. The details of the architecture are not reported in this section and only the arguments at the basis of the choice of the input variables and results are discussed. The input variables are selected as the inlet total pressure ($p_t$) and temperature ($T_t$), which define the operating conditions of the turbine. To account for the unsteady deviations of the turbine, the first time derivative of the total pressure pulse ($\partial p_t / \partial t$) is used as an input variable since deviations have been shown to be proportional to the first derivative of the pressure pulse in Section 4.3.3. Moreover, the inclusion of the time derivative of the total pressure pulse embeds the temporal information related to the pulse shape and guarantees the surjectivity between input and output variables. All the input variables are defined upstream of the turbine and usually constitute the boundary conditions that are imposed at the inlet of the system in CFD numerical set-ups. Consequently, such quantities are considered to be known a-priori, e.g. generated from a reduced-order model or measured experimentally. As output, the model returns the instantaneous turbine torque ($\tau$) and the circumferentially-averaged relative inflow angle ($\beta$).
Table 4.2: Summary of the different organizations of the training, cross-validation, and test data sets for the different cases analyzed.

<table>
<thead>
<tr>
<th></th>
<th>Training</th>
<th>Validation</th>
<th>Test</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case A</td>
<td>Amplitude</td>
<td>$\phi = 0.4, 1.0, 1.6$</td>
<td>$\phi = 0.7$</td>
</tr>
<tr>
<td></td>
<td>Frequency</td>
<td>$\phi = 0.7$</td>
<td>$\phi = 1.3$</td>
</tr>
<tr>
<td>Case B</td>
<td>Amplitude</td>
<td>$\phi = 0.4, 1.0, 1.3$</td>
<td>$\phi = 0.7$</td>
</tr>
<tr>
<td></td>
<td>Frequency</td>
<td>$\phi = 0.7$</td>
<td>$\phi = 1.6$</td>
</tr>
<tr>
<td>Case C</td>
<td>Amplitude</td>
<td>$\phi = 0.4, 1.0, 1.3, 1.6$</td>
<td>$\phi = 0.7$</td>
</tr>
<tr>
<td></td>
<td>Frequency</td>
<td>$f = 20, 60, 80, 100$ Hz</td>
<td>$f = 40$ Hz</td>
</tr>
</tbody>
</table>

The data set used to train and validate the model is constituted by the results presented in Section 4.3.1 and Paper 3. The data set has been divided into three sub-datasets following a standard procedure for the training and validation of machine learning models. The training data set is used during the training phase to adjust the weights of the different neurons forming the architecture. At the same time, the model error is monitored on a cross-validation data set to avoid overfitting and underfitting. Lastly, the accuracy of the model is assessed on a test data set. In total, three different cases are investigated with different compositions of the training data set. The organization of the training, validation, and test data sets is summarized in Tab.4.2. For Case A and Case B, only variations of the pulse amplitude are included in the data set. For Case A, the training data set is constituted of cases $\phi = 0.4, 1.0, 1.6$, while the validation and test data sets are represented by cases $\phi = 0.7$ and $\phi = 1.3$, respectively. The inclusion of cases $\phi = 0.4$ and $\phi = 1.6$ in the training data set is meant to test the interpolation capacity of the NN model for a pulse shape that is contained within the bounds of the training data set. On contrary, the extrapolation capacity of the NN model is tested in Case B, where the test data set is constituted by case $\phi = 1.6$, which is outside the bounds of the training data set. Lastly, for Case C, variations of the pulse frequency are included to test the capacity of the NN model to predict the turbine performance for a more general pulse shape.

In Fig.4.20, the instantaneous turbine torque and circumferentially-averaged relative inflow angle predicted by the CFD and NN models are reported respectively. For Case A, the NN model shows an excellent capability in predicting both the instantaneous turbine torque (Fig.4.20a) and circumferentially-averaged relative inflow angle (Fig.4.20b). The RMSE is reported equal to 0.014 and 1.07 for the turbine torque and relative inflow angle. For Case B, the prediction of the NN deteriorates, as expected, at the minimum and maximum of the pulse. However, even in this case, the agreement between CFD and NN results can be considered satisfactory since the RMSE is reported equal to 0.023 and 2.51 for the turbine torque and relative inflow angle, respectively. The results
Figure 4.20: Instantaneous turbine torque and circumferentially-averaged relative inflow angle predicted by the CFD and NN models for Case A (Fig.4.20a and 4.20b), Case B (Fig.4.20c and 4.20d), and Case C (Fig.4.20e and 4.20f), respectively.

show a good capacity of the NN to extrapolate the performance for (reasonable) conditions outside the bounds of the training data set. As final test, for Case C, frequency variations are included in the data set. The results predicted by the NN show a good capacity of the NN model to predict the turbine performance for a more general pulse shape with a great degree of accuracy. Indeed, the RMSE improves as compared to Case B and is reported equal to 0.018 and 2.00 for the turbine torque and relative inflow angle, respectively.
4. Summary of results

The large majority of reduced-order models for turbocharger turbines are based on 0D or 1D models which aim to approximate complex physics phenomena through geometrical and physics simplifications. On the other hand, data-driven methods make use of high-fidelity experimental or numerical data for regression modeling, overcoming the limitations caused by physics simplifications. However, this approach is rarely adopted in turbocharger modeling due to its larger computational cost. In this section, we have demonstrated that data-driven methods, in the form of a fully-connected neural network, are a suitable approach for modeling the turbocharger radial turbine by showing a comparable, or even greater, accuracy of such models with respect to other 0D or 1D models presented in the literature.

4.3.5. Pulse optimization

As exposed in the introduction in Section 1.3, most of the works attempting to characterize the turbine performance in pulsating flow conditions proceed through independent and discrete variations of the pulse characteristics such as amplitude and frequency. This approach, despite providing useful information about the characterization of the turbine performance, can be difficult to employ to find the optimum working conditions of the turbine in terms of pulse shape because it is based on complex and time-consuming CFD simulations or experimental campaigns. To answer the need of identifying the optimum pulse shape, in Paper 5, a new algorithm based on a gradient-descent method is developed targeting the maximization of the time-averaged turbine torque given different types and numbers of constraints. For the efficient evaluation of the turbine performance, the gradient-based algorithm is coupled with a similar NN model as the one presented in Section 4.3.4 and Paper 4.

The optimization problem is formulated as a minimization problem of a certain cost function $J(\vec{x})$ subject to a series of constraint $g_i(\vec{x})$ as in Eq.4.6.

$$\min_{\vec{x}} J(\vec{x}), \quad \text{subject to } g_i(\vec{x}) = 0, \quad i = 1, \ldots, N \quad (4.6)$$

Here, $N$ represents the number of constraints imposed. For the algorithm implementation, we have employed the SciPy implementation of the Sequential Least SQuares Programming (SLSQP) method (Virtanen et al. 2020). Physically, we are interested in finding the optimal total pressure pulse $p_t$ at the inlet that yields the highest time-averaged turbine torque $\tau$, given a set of constraints. In the algorithm, the turbine torque is modeled by a NN model similar to the one presented in Section 4.3.4. The NN takes as input variables the total pressure and temperature pulses and the first and second time derivatives of the total pressure pulse. As outputs, the NN model returns the turbine torque $\tau$ and the Mach number (at the inlet of the system) as in Eq.4.7.

$$f_{NN} (p_t, T_t, \partial_t p_t, \partial_t^2 p_t) = \begin{pmatrix} \tau \\ Ma \end{pmatrix} \quad (4.7)$$
Consequently, without considering any constraint, it is possible to define the optimization problem as

$$\min_{p_t} \left( -\bar{\tau} \right) = \min_{p_t} \left( -\bar{f}_{NNN} (p_t, T_t, \partial_t p_t, \partial_t^2 p_t) \right), \quad (4.8)$$

with the bar indicating the time average operation and the superscript indicating the first components of $f_{NNN}$. The total pressure pulse function $p_t$ is parametrized in terms of a first-order Fourier expansion:

$$p_t(t) = \vartheta_0 + \vartheta_1 \cos(\omega t) + \vartheta_2 \sin(\omega t). \quad (4.9)$$

In this expansion, $\omega$ represents the angular frequency of the pressure pulse and is kept constant during the optimization. Consequently, the gradient-driven modification of $p_t$ is achieved by varying the set of parameters $\Theta = \{\vartheta_0, \vartheta_1, \vartheta_2\}$. The choice of truncating the Fourier series to the first order is dictated by the need of fixing the frequency of the pulse and to not allow higher order harmonics to appear in the signal. The total temperature pulse necessary as input for the neural network model is calculated based on the isentropic relation $p_t^{1 - \gamma} = \text{const}$. For the analysis, two different types of constraints are considered. In the first case, the time-averaged total pressure is kept the same as the initial one for each modification of the total pressure pulse evaluated by the algorithm. In the second case, the time-averaged mass flow rate is kept the same instead of the total pressure pulse. The mass flow rate at the inlet boundary is calculated based on the compressible flow equations in Eq.4.10.

$$\dot{m} = \frac{A \sqrt{T_t}}{\sqrt{\gamma} M a} \left( 1 + \frac{\gamma - 1}{2} M a \right)^{-\frac{\gamma + 1}{2(\gamma - 1)}} \quad (4.10)$$

Here, $A$ represents the area of the inlet section. The calculation of the mass flow rate through Eq.4.10 requires the evaluation of the Mach number at the inlet. This is achieved through the NN model which returns the Mach number at the inlet section as an output variable together with the turbine torque. The optimization procedure is reported in the pseudo Algorithm 1.

In Fig.4.21a, the comparison between the initial and optimal pulses is reported for the constraint applied on the total pressure. It is worth noticing that the optimal pulse tends towards a more flat profile characterized by a lower pulse amplitude than the initial one. The effects of the optimization are shown on the turbine torque predicted by the NN model in Fig.4.21b. From the initial to the optimal pulse shape, the minimum and maximum values of the turbine torque vary by $+15\%$ and $-6.4\%$, respectively. The optimization algorithm leads to a limited improvement of the time-averaged turbine torque of $0.5\%$, which suggests that the initial pulse was already close to the optimal one. In Fig.4.21c, the comparison between the initial pulse and the optimal pulses is reported for constraint applied on the mass flow rate. Surprisingly and differently from the previous case, the optimal pulse shape tends to be the one characterized by the highest possible pulse peak allowed by the constraints.
Algorithm 1: Gradient-based algorithm

Initial pulse $p_t$

while $\left| \frac{\partial J}{\partial \vartheta_i} \right| < \varepsilon$ do

Compute total temperature

$T_t \leftarrow p_t^{1-\gamma} T_t^\gamma = \text{const}$

Compute turbine torque and Mach number

$\tau, Ma \leftarrow p_t, T_t, \frac{\partial p_t}{\partial t}, \frac{\partial^2 p_t}{\partial t^2}$

Compute mass flow rate for constraint

$m \leftarrow \frac{A_p_t}{\sqrt{T_t}} \sqrt{\frac{\gamma}{R}} Ma \left( 1 + \frac{\gamma - 1}{2} Ma \right)^{-\frac{\gamma+1}{2(\gamma-1)}}$

Compute gradient of the cost function

$\frac{\partial J}{\partial \vartheta_i}$

Update total pressure pulse

$p_t(t) = \vartheta_0 + \vartheta_1 \cos(\omega t) + \vartheta_2 \sin(\omega t)$

end

imposed. The instantaneous turbine torque for the initial and optimal pulse shown in Fig.4.21d reflects well the trend of the total pressure pulse. Indeed, the minimum turbine torque decreases by 24%, the maximum turbine torque increases by 27%, while the average turbine torque increases by 11% from the initial to the optimal pulse. The improvement of the turbine performance can be related to the higher energy content that characterizes the flow for the optimal pulse. Indeed, since no constraint is imposed on the average value, the time-averaged total pressure increases by 6% with respect to the initial pulse. As a consequence, it can be inferred that a pulse characterized by a larger mass
4.3. Performance analysis

Figure 4.21: Comparison of the initial and optimal total pressure pulses (left) and turbine torque (right) predicted by the NN model for the constraints on the total pressure (Fig.4.21a and 4.21b) and mass flow rate (Fig.4.21c and 4.21d).

The results presented confirm the considerations expressed in Section 4.3.2 regarding the effects of the type of constraints on the turbine performance.

4.3.6. Exergy-based analysis

In the context of pulse parametric studies, the importance of the constraints on the pulse shape has been shown to significantly affect the performance trends with respect to variations of the pulse characteristics. In particular, by varying the pulse characteristics, i.e. amplitude and frequency, the energy content of the flow varies, and so does the maximum work that the turbine can extract from the flow. Such limit is expressed by the concept of exergy, which is a composite variable that represents the maximum amount of shaft work (per unit mass) that can be extracted until the fluid particles reach an equilibrium state with the environment, usually referred to as dead state. The specific flow exergy is defined in Eq.4.11.

\[
e_f = h_t(T_t) - h_o - T_o \left[ s(T_t, p_t) - s_o \right], \tag{4.11}
\]
4. Summary of results

Here, \( h \) and \( s \) represent the specific enthalpy and entropy while the subscripts \( t \) and \( o \) refer to the total conditions and the dead state, respectively. Exergy depends on both the properties of the flow and the environment the fluid interacts with. In this thesis, the dead is chosen as the ambient conditions since we want to investigate the turbine performance from a global point of view (system + environment) and not only from the system perspective.

In order to appropriately compare the effects of different pulse shapes on the system performance, in Paper 6 and Paper 7, the cycle-averaged flow exergy is imposed as a constraint for the different pulse shapes investigated. In this way, the turbine performance are not influenced by the different amount of work available in the flow and the pulse characteristics can be compared in absolute terms. Such constraint is obtained by uniformly scaling the total temperature pulse while different pulse characteristics (amplitude, frequency, and temporal gradient) are varied on the mass flow rate pulse. With respect to previous works, closer engine-like conditions are considered. The inlet pipe preceding the volute is substituted by the exhaust manifold (system B in Fig.2.4) and heat transfer effects are considered by defining the static temperature on the system walls.

For the analysis of the system performance and aerothermodynamic losses in the system, an exergy-based post-processing approach is adopted by considering the exergy transport equation expressed in integral form in Eq.4.12.

\[
\frac{d}{dt} \left[ \iiint_{V^*(t)} (\rho e_f) \, dV \right] + \iiint_{S^*(t)} \left[ \rho e_f \left( \vec{u} - \vec{u}_b \right) \cdot \vec{n} \right] \, dS =
\]

\[
- \iiint_{S^*(t)} \left[ \left( \vec{r} \times \vec{f}_{net} \right) \cdot \vec{n} \right] \, dS - \iiint_{S^*(t)} \left[ \left( 1 - \frac{T_o}{T_w} \right) (\vec{q} \cdot \vec{n}) \right] \, dS
\]

\[
- T_o \dot{S}_{gen} + \frac{d}{dt} \left[ \iiint_{V^*(t)} p \, dV \right].
\]

Here, the term \( A \) represents the exergy accumulation in the control volume, \( B \) the net exergy flux across the surfaces of the control volume, \( C \) the turbine power, \( D \) the exergy gained/lost due to the heat transfer across the surface, \( E \) the exergy dissipated by internal irreversibilities, and \( F \) the unsteady contribution of the pressure. The internal irreversibilities can be computed by rearranging Eq.4.12 as \( E = -B - C - D - (A - F) \) and by numerically evaluating the terms on the RHS. Such an approach is called the indirect method (Herwig & Kock 2007). However, in this analysis, we are interested in distinguishing between the different type of aerothermodynamic (thermal and viscous) losses. As a
consequence, a direct method is adopted and the viscous and thermal losses are
computed through Eq.4.13 and 4.14, respectively.

\[ \dot{S}_{gen,\text{viscous}} = \iiint_{V^*(t)} \left[ \frac{\mu_{eff}}{T} \left( 2 S_{ij} S_{ij} - \frac{2}{3} S_{kk} S_{kk} \right) \right] dV \]  

(4.13)

\[ \dot{S}_{gen,\text{thermal}} = \iiint_{V^*(t)} \left[ \frac{k_{eff}}{T^2} \left( \frac{\partial T}{\partial x_j} \right)^2 \right] dV \]  

(4.14)

Here, \( \mu_{eff} \) and \( k_{eff} \) are the effective dynamic viscosity and thermal conductivity,
respectively, the sum of the molecular and turbulent ones. The relative impor-
tance between the viscous and thermal irreversibilities is compared through the
Bejan number (see Eq.4.15) which ranges from 0, when the viscous irreversibili-
ties are dominant, to 1, when the thermal irreversibilities are much larger than
the viscous ones.

\[ Be = \frac{1}{1 + \frac{\dot{S}_{gen,\text{viscous}}}{\dot{S}_{gen,\text{thermal}}}} \]  

(4.15)

In the following, we focus on the analysis of the pulse amplitude effects
by comparing two different pulse shapes characterized by a 5% difference in
the pulse amplitude. These results represent a summary of Paper 6 while, in
Paper 7, frequency and temporal gradient effects are evaluated based on the
above-mentioned approach. The lower amplitude pulse is referred to with the
nomenclature EVS1 (exhaust valve strategy) while the higher amplitude pulse
by EVS2. In Fig.4.22, the exergy budget and Bejan number are compared for
the different components of the hot-side system between EVS1 and EVS2 while
the quantitative data are summarized in Tab.4.3. The main result from the
exergy budget analysis is represented by the increment of the turbine power
as pulse amplitude increases. Although Fig.4.22a and Tab.4.3 highlight the
turbine power as the least sensitive exergy budget term among all, a +1.3% percentage difference is still significant in terms of turbine performance. The
growth of the turbine power is even more significant when considering that,
at the turbine inlet, the inflow exergy is smaller for EVS2 compared to EVS1.
Such results confirm the trends seen in previous sections regarding the influence
of the pulse amplitude on the turbine performance.

The exergy lost by heat transfer accounts for \( \approx 5\% \) of the cycle-averaged
inflow exergy, with the volute responsible for \( \approx 65\% \) of the total. As reported
in Fig.4.22 and Tab.4.3, the exergy lost by heat transfer shows an ascending
trend with increasing pulse amplitude in both the exhaust manifold and volute.
Percentage differences are reported equal to +13.3% and +1.8% and demonstrate
a significant influence of the pulse amplitude on heat transfer, in particular at
the exhaust manifold. The higher sensitivity of the heat transfer at the exhaust
manifold, equal to +13.3%, derives from the stronger influence of the boundary
4. Summary of results

Table 4.3: Cycle-averaged values of the turbine power (term C), exergy lost by heat transfer (term D) and exergy losses via total internal irreversibilities (term E) at the exhaust manifold, volute, and rotor, respectively. Viscous and thermal irreversibilities for the entire system are also reported.

<table>
<thead>
<tr>
<th>Component</th>
<th>Manifold</th>
<th>Volute</th>
<th>Rotor</th>
<th>Irreversibilities</th>
</tr>
</thead>
<tbody>
<tr>
<td>EVS1</td>
<td>637</td>
<td>193</td>
<td>1191</td>
<td>330</td>
</tr>
<tr>
<td>EVS2</td>
<td>722</td>
<td>197</td>
<td>1213</td>
<td>346</td>
</tr>
<tr>
<td>(\Delta \phi) [%]</td>
<td>+13.3%</td>
<td>+2.1%</td>
<td>+1.8%</td>
<td>+4.8%</td>
</tr>
<tr>
<td>(T_o \dot{S}_{gen,viscous}) [W]</td>
<td>4455</td>
<td>167</td>
<td>4512</td>
<td>171</td>
</tr>
<tr>
<td>(T_o \dot{S}_{gen,thermal}) [W]</td>
<td>471</td>
<td>219</td>
<td>505</td>
<td>209</td>
</tr>
</tbody>
</table>

Figure 4.22: Pulse amplitude effects on the exergy budget (Fig.4.22a) and Bejan number (Fig.4.22b) on the different components of the system: exhaust manifold, volute, and rotor. Exergy budget data are normalized by the cycle-averaged inflow exergy, \(\dot{m} e_f\), at the exhaust manifold inlet.

conditions at the inlet of the system. Indeed, the temperature difference between the fluid and the walls decreases as the fluid particles travel through the different components of the system. As a consequence, the temperature difference between the two EVSs is larger at the exhaust manifold inlet than at the exhaust manifold-volute interface. In Fig.4.23, the time-varying exergy lost by heat transfer at the exhaust manifold and volute is plotted over the pulse cycle. It can be noticed how the heat transfer rate is larger at the pulse peak for EVS2 compared to EVS1, highlighting a remarkable influence of the pulse amplitude on the heat transfer. Moreover, during the intra-pulse phase, it is interesting to notice that heat transfer is reversed, directed from the surroundings to the domain.

The total internal irreversibilities, the sum of the viscous and thermal ones, account for a small but significant fraction, \(\approx 2\%\), of the cycle-averaged inflow
exergy. With increasing pulse amplitude, the entropy generation increases in all the hot-side components (see Fig.4.22a and Tab.4.3) with the largest variation, equal to +5%, occurring inside the volute, which is responsible for the largest fraction, ≈ 50%, of the total internal irreversibilities. A deeper understanding of the nature of the irreversibilities is given by the Bejan number (see Fig.4.22b) which decreases in all the hot-side components as the pulse amplitude increases. This trend highlights that entropy generation associated with viscous mechanisms increases compared to thermal irreversibilities. This is justified by the fact that viscous effects, proportional to the velocity gradients, are larger with increasing pulse amplitude and maximum velocities. Inside the rotor, the Bejan number is characterized by smaller values than in the exhaust manifold and volute because the flow field is dominated by the viscous effects, i.e. larger velocity gradients caused by the high rotational speed of the rotor. The temporal evolution of the exergy losses caused by viscous entropy generation is reported in Fig.4.24 over the pulse cycle. Although the trend looks similar for both EVSs during the pulse phase, EVS2 is characterized by a higher peak, where the percentage variation is reported equal to +8.3% showing a remarkable influence of the pulse amplitude. Globally, viscous irreversibilities increase by +7.2% from EVS1 to EVS2 (see Tab.4.3) while percentage differences in the different components of the hot-side system are reported equal to +10.3%, +8.2%, and +3.4% for the exhaust manifold, volute, and rotor, respectively. As for the exergy losses caused by heat transfer, it is worth noticing that the effects of the boundary conditions reduce as the flow travels through the different components of the system. As reported in Tab.4.3, the exergy losses caused by thermal irreversibilities decrease with increasing pulse amplitude from EVS1 to EVS2. Despite this trend, the evolution in time of the thermal irreversibilities, given in Fig.4.25, shows that EVS2 is characterized by a higher peak than EVS1.

Figure 4.23: Time-varying exergy lost by heat transfer over the pulse cycle inside the exhaust manifold and volute.
Figure 4.24: Time-varying exergy loss via viscous mechanisms over the pulse cycle.

Figure 4.25: Time-varying exergy loss via thermal mechanisms over the pulse cycle.

with a percentage difference equal to +12.1% at the pulse peak. Moreover, as already observed by the Bejan number in Fig.4.22b, the exergy losses caused by thermal irreversibilities are particularly small inside the rotor, where they account for only \( \approx 6\% \) of the total.

The exergy budget methodology used shows an interesting aspect in terms of global performance. Indeed, only \( \approx 13\% \) of the total available exergy is converted into work by the radial turbine. As a consequence, the largest part of the flow exergy is neither converted into work nor dissipated by internal and external mechanisms. Consequently, \( \approx 80\% \) of the initial exergy is still available.
downstream of the turbine. This observation suggests that more power can be extracted from the exhaust gases. This result can be achieved by adopting more complex turbine systems characterized by additional stages.
Conclusions and outlook

The present thesis investigates the effects of pulsating flow conditions on a radial turbine meant for turbocharging applications. The research intends to characterize the performance of the turbine subject to on-engine conditions by exploring the sensitivity of the system to pulse characteristics. Such knowledge is then used to develop reduced-order models for the turbine.

The research output of the thesis is represented by seven papers, fully reproduced in the following. The works can be divided into two categories. **Paper 1** and **Paper 2** investigate the flow characteristics in different components of the hot-side, i.e. volute and rotor, in both continuous and pulsating flow conditions. **In Paper 3-7**, the turbine response to different pulse characteristics is investigated by different approaches. In the following, the main achievements and limitations are summarized and possible extensions of the results are suggested for each specific paper.

**Paper 1:**

- **Achievements:** The volute-rotor interaction in continuous flow conditions is investigated using Large Eddy Simulations. The results demonstrate that the non-uniform inflow conditions at the volute-rotor interface significantly affect the blade performance in the circumferential direction. Specifically, a significant improvement of the turbine torque is reported in the region affected by the wake generated by the volute tongue. Despite the volute-rotor unsteady interaction is neglected in steady-state models, a steady-state RANS model using the mixing-plane approach for the treatment of the stator-rotor interface is shown to accurately predict the averaged performance of the turbine as compared to an LES approach. Consequently, the computational cost of investigating the averaged performance can be reduced by using steady-state RANS models.

- **Limitations & Outlook:** Due to the high computational cost of Large Eddy Simulations, the analysis is limited to an operating point characterized by a low rotational speed and expansion ratio. To generalize the flow characteristics observed, further operating points should be investigated at higher rotational speeds and expansion ratios.
Paper 2:

- **Achievements:** The turbine performance and vortical structures developing in the blade passage are studied using Large Eddy Simulations. The results show that radial turbines operating in pulsating flow conditions are characterized by lower isentropic efficiencies as compared to corresponding continuous flow conditions. This trend is caused by the flow separations that develop on the blade pressure and suction sides at extreme off-design conditions. As a consequence, during pulsating flow conditions, the flow expansion through the turbine departs more from an ideal isentropic process as compared to corresponding continuous flow conditions.

- **Limitations & Outlook:** The approach used in this work is limited to the correlation between turbine performance indices and a qualitative analysis of the flow features in the blade passage. Due to the unsteady nature of the boundary conditions, the characterization of the more dominant flow structures in pulsating flow conditions is a challenging goal. Indeed, statistics acquisition is impractical for such kind of applications since phase-averaging is required for each instant of the pulse. To overcome this limitation, a quasi-steady approach should be preferred. This choice would enable the application of mode decomposition techniques, such as Proper Orthogonal Decomposition (POD) or Dynamic Mode Decomposition (DMD), to quantitatively characterize the more dominant flow structures.

Paper 3:

- **Achievements:** Using an unsteady RANS model, a classic parametric study is performed to assess the response of the turbine performance to pulse amplitude and frequency variations. Characteristic trends of the turbine performance are highlighted. In particular, the power output of the turbine improves by increasing pulse amplitude and decreasing pulse frequency. Interestingly, the increase in the turbine power is associated with a drop in the isentropic efficiency. This trend can be related to the range of relative inflow angles experienced by the turbine. As the range expands, the turbine is exposed to more extreme events, e.g. flow separations on the blade pressure and suction sides, which cause the expansion to depart more from an ideal isentropic process. The nature of the unsteady turbine performance deviations from quasi-steady conditions is investigated. It is found that the intensity of the deviations increases with increasing pulse amplitude and frequency. However, through a simple algebraic model, it is demonstrated that the performance deviations are proportional to the normalized reduced frequency, which is independent of the pulse amplitude and frequency. As a consequence, the performance deviations are proportional to the time derivative of the pressure pulse and it is considered inappropriate to correlate them to variations in the pulse amplitude and frequency.
5. Conclusions and outlook

- **Limitations & Outlook:** The study was limited to operating points characterized by the same rotational speed of the turbine, whose effects on the performance deviations from quasi-steady to pulsating flow conditions should be investigated. Moreover, the algebraic model was tested for a single speed line, so the range of applicability of the model has not been investigated for variations in the rotational speed of the turbine.

**Paper 4:**

- **Achievements:** A data-driven method is developed to model the radial turbine performance under pulsating flow conditions. The model, based on a fully-connected neural network, returns as outputs the instantaneous turbine torque and relative inflow angle given, as inputs, a minimum number of variables defined upstream of the turbine system. To account for the wave propagation from the inlet of the system to the rotor, the neural network model is trained with data obtained in pulsating flow conditions. The comparison between CFD and NN predictions shows a great capacity of the neural network to model the unsteady performance of the turbine subject to pulsating flow conditions. The model is shown to make accurate predictions for pulse conditions within and outside the bounds of the training data. The model developed represents a valid alternative to the more common 0D and 1D reduced-order models used for this kind of application.

- **Limitations & Outlook:** Similarly to the previous work, the analysis is limited to a single rotational speed of the turbine while heat transfer effects are neglected. The inclusion of such phenomena is essential to model the turbine performance under real on-engine conditions. Moreover, to assess the applicability of similar NN models at the industrial level, it is important to estimate the minimum size of the training data set that guarantees an acceptable accuracy for the turbine performance predictions in the entire range of operating conditions of the turbine.

**Paper 5:**

- **Achievements:** A gradient-based optimization algorithm, which targets the definition of the optimum total pressure pulse shape that maximizes the time-averaged turbine torque, is developed and presented. After the validation of the algorithm, the effects of different types of constraints on the optimum pulse are investigated. Results show that the type of constraint has a significant effect on the final optimum pulse shape. In particular, for an equality constraint on the time-averaged total pressure, the optimum pulse tends towards a more flat profile. Conversely, for an equality constraint on the time-averaged mass flow rate, the optimum pulse shape tends towards a narrower shape characterized by a higher pulse amplitude than the initial pulse. These results are of particular interest in the context of numerical pulse parametric studies since the
choice of the type of inlet boundary conditions determines the type of constraint imposed.

- **Limitations & Outlook:** Since the pulse shape is discretized by a first-order Fourier series, variations of the pulse frequency are not allowed. Moreover, the study is limited to a single turbine rotational speed. The inclusions of such parameters should be investigated in future works to generalize the results observed.

**Paper 6-7:**

- **Achievements:** A parametric study of the hot-side system response to variations of the pulse amplitude, frequency, and temporal gradient is carried out. Since different pulse strategies are characterized by different energy contents, the performance trends of the turbine are influenced by the maximum work available in the flow. Such a limit is represented by the concept of exergy. As a consequence, to enable a direct comparison for each pulse characteristic, the different pulses are designed by imposing a constraint on the time-averaged inflow exergy. An exergy-based post-processing approach has enabled the investigation of the hot-side response from a global (system + environment) point of view. Results demonstrate that only 20% of the total exergy is converted into work or dissipated by internal and external mechanisms. This observation suggests the utilization of a second turbine stage to exploit the residual energy contained in the exhaust gases.

- **Limitations & Outlook:** The simplification of the boundary conditions represents the main limitation of the heat transfer analysis of this work. In the future, to minimize the uncertainties related to the thermal boundary conditions, Conjugate Heat Transfer (CHT) will be performed.
Acknowledgements

This thesis belongs to the many colleagues, friends, and family members that have unconditionally supported me over the past four years.

I am deeply grateful to my main supervisor, Mihai Mihaescu, for his dedication to my research project and the freedom given. I cannot imagine another situation where I could have expressed myself more freely. A special thanks goes to my co-advisor, Anders Dahlkild, for his kindness and patience in explaining to me concepts of thermodynamics (and also the complex bureaucratic rules of the department when needed). I also want to thank Ardershir Hanifi for the internal review of my thesis, the administrative staff for their support, Emelie and Frida for their unmatchable dedication to writing the Swedish abstract, and Francesco and Thomas for helping me during the final review of the thesis when I was not able to read even a road sign. I am immensely grateful for the collaboration with two great researchers as Marco and Shyang. The fruitful discussions and your contagious enthusiasm have encouraged me to do better every day. I would like to thank Thomas Biesinger from BorgWarner for his precious support.

This journey would not have been the same without all the new friends met along its path. A huge thanks goes to my first office mates and friends, Lukas and Niclas, for the heroic nights spent together starting from our office and ending who knows where. I express my immense gratitude to Francesco and Federico who put up with my complaints and the stories of the tragicomic events of my turbulent life. I want to thank all my office mates Emelie, Thomas, Louis, Frida, Peng, and Elias, and to apologize to Emelie and Frida for my poor Swedish level. I would like to extend my sincere thanks to all the people making our department such an enjoyable and fascinating place. ”Grazie” to all the Italian friends in the department that make me feel less far from home. Thank you, Oscar, Einar, and Emilio for the unforgettable nights spent together in Lappis. A special greeting goes to the fantacalcio friends. The victory of the league represents by far the greatest achievement of this year; the Ph.D. has been only an instrument to develop the necessary analytical skills to triumph in the league.
Thank you, Flavia for enriching my life every day even with the smallest acts. None of what I have accomplished would have been possible without the love and support of my parents. To them, I dedicate my heartfelt gratitude.

This doctoral thesis project was conducted within the framework of the Competence Center for Gas Exchange (CCGEx) at KTH. Support from Scania AB, Volvo GTT, BorgWarner Turbo Systems, Volvo Cars/Aurobay, Converge Sci, Wärtsilä, and the Swedish Energy Agency is greatly acknowledged. The Swedish National Infrastructure for Computing (SNIC) via Parallel Computing Center (PDC) at KTH, the National Supercomputer Centre (NSC) at LiU and the DECI resource Mahti based in Finland at CSC with support from the Partnership for Advanced Computing in Europe (PRACE) aisbl are also acknowledged for providing valuable computational resources.
Bibliography


EPA 2022 Inventory of u.s. greenhouse gas emissions and sinks. https://www.epa.


Kolmogorov, A. N. 1941 The local structure of turbulence in incompressible viscous fluid for very large reynolds numbers. Cr Acad. Sci. URSS 30, 301–305.


Bibliography


RAJOO, S., ROMAGNOLI, A. & MARTINEZ-BOTAS, R. F. 2012 Unsteady performance


**Reynolds, O.** 1883 Xxix. an experimental investigation of the circumstances which determine whether the motion of water shall be direct or sinuous, and of the law of resistance in parallel channels. *Philosophical Transactions of the Royal society of London* (174), 935–982.


**Richardson, L. F.** 1922 *Weather prediction by numerical process*. University Press.


Bibliography 69


Part II

Papers
Summary of the papers

Paper 1

*Analysis of the volute-rotor interaction in radial turbines by means of large eddy simulations*

Large eddy simulations are used to investigate the volute-rotor interaction in a nozzleless radial turbine operating in continuous flow conditions. Due to the volute tongue wake, the inflow conditions at the volute-rotor interface are highly non-uniform. As a consequence, the blade torque significantly varies in the circumferential direction. The effects on the turbine performance predictions of modeling choices are also investigated by comparing a steady-state RANS model and an LES approach.

Paper 2

*Large eddy simulations of a turbocharger radial turbine under pulsating flow conditions*

The flow dynamics in the rotor passage of a turbocharger radial turbine operating in pulsating flow conditions is investigated using large eddy simulations. The turbine experiences a drop in the isentropic efficiency as compared to corresponding continuous flow conditions. This trend is the result of the off-design inflow conditions the turbine blade is exposed to when operating in pulsating flow conditions. At low mass flow rates, the flow in the blade passage is dominated by a flow separation on the blade pressure side. As the mass flow rate increases, the separation becomes smaller and moves to the suction side for high mass flow rate conditions. As a consequence of the rich variety of secondary flows developing in pulsating flow conditions, the expansion through the turbine departs more from an ideal isentropic process.

Paper 3

*Assessment of the unsteady performance of a turbocharger radial turbine under pulsating flow conditions: parametric study and modeling*

A classic parametric study is carried out to investigate the turbine performance response to variations in the pulse amplitude and frequency. The computational cost of the study is reduced by using an experimentally-validated unsteady
RANS model. Characteristic trends of the turbine performance are highlighted for variations of the pulse amplitude and frequency. Moreover, an algebraic model for predicting the turbine performance deviations from quasi-steady to pulsating flow conditions is developed and presented.

**Paper 4**

*Modeling radial turbine performance under pulsating flow by machine learning method*

In this work, a reduced-order model based on a fully-connected neural network architecture is developed to predict the unsteady performance of a turbocharger radial turbine. The model, trained with numerical data obtained by an experimentally-validated unsteady RANS model in pulsating flow conditions, is shown to accurately predict the instantaneous turbine torque and relative inflow angle. The model developed represents a valid alternative to more common 0D or 1D reduced-order models.

**Paper 5**

*Gradient-based optimization of pulsating inflow conditions for turbocharger radial turbines*

The work presents the development and application of a gradient-based optimization algorithm for identifying the turbine optimum working conditions in terms of total pressure pulse shape. The algorithm searches for the optimum total pressure pulse shape that maximizes the time-averaged turbine torque. The influence on the optimum pulse shape of equality constraints on the time-averaged total pressure and mass flow rate are investigated.

**Paper 6**

*Turbocharger radial turbine response to pulse amplitude*

In this work, the hot-side system response is investigated for pulse amplitude variations using an exergy-based post-processing approach. The pulses are designed by imposing a constraint on the time-averaged inflow exergy. In this way, the behavior of the turbine is not affected by a different maximum work that can be extracted from the flow.

**Paper 7**

*Influence of pulse characteristics on turbocharger radial turbine*

The hot-side system response is studied for variations in the pulse amplitude, frequency, and temporal gradient. The work intends to investigate the influence of pulse characteristics on the aerothermodynamic losses in the system under on-engine conditions. For this reason, heat transfer effects are considered and the computational model incorporates the exhaust manifold to account for the secondary flows that develop in real on-engine applications. The analysis
makes use of an exergy-based post-processing approach to assess the system performance from a global system perspective.