Centrifugal compressor flow instabilities at low mass flow rate

by

Elias Sundström

TRITA-MEK Technical report 2016:06
ISSN 0348-467X
ISRN KTH/MEK/TR–16/06–SE

©Elias Sundström 2016
Centrifugal compressor flow instabilities at low mass flow rate

Elias Sundström 2016, CCGEx and Linné Flow Centre, KTH Mechanics, Kungliga Tekniska Högskolan, SE-100 44 Stockholm, Sweden

Abstract

Turbochargers play an important role in increasing the energetic efficiency and reducing emissions of modern power-train systems based on downsized reciprocating internal combustion engines (ICE). The centrifugal compressor in turbochargers is limited at off-design operating conditions by the inception of flow instabilities causing rotating stall and surge. They occur at reduced engine speeds (low mass flow rates), i.e. typical operating conditions for a better engine fuel economy, harming ICEs efficiency. Moreover, unwanted unsteady pressure loads within the compressor are induced; thereby lowering the compressors operating life-time. Amplified noise and vibration are also generated, resulting in a notable discomfort.

The thesis aims for a physics-based understanding of flow instabilities and the surge inception phenomena using numerical methods. Such knowledge may permit developing viable surge control technologies that will allow turbochargers to operate safer and more silent over a broader operating range. Therefore, broadband turbulent enabled compressible Large Eddy Simulation (LES) calculations have been performed and several flow-driven instabilities have been captured under unstable conditions. LES produces large amounts of 3D data which has been post-processed using Fourier spectra and Dynamic Mode Decomposition (DMD). These techniques are able to quantify modes in the flow field by extracting large coherent flow structures and characterize their relative contribution to the total fluctuation energy at associated. Among the main findings, a dominant mode was found which describes the filling and emptying process during surge. A narrowband feature at half of the rotating order was identified to correspond to co-rotating vortices upstream of the impeller face as well as elevated velocity magnitude regions propagating tangentially in the diffuser and the volute. Dominant mode shapes were also found at the rotating order frequency and its harmonics, which manifest as a spinning mode shape localized at the diffuser inlet.

From the compressible LES flow solution one can extract the acoustic information and the noise affiliated with the compressor. This enable through data correlation quantifying the flow-acoustics coupling phenomena in the compressor. Detailed comparison of flow (pressure, velocity) and aeroacoustics (sound pressure levels) predictions in terms of time-averaged, fluctuating quantities, and spectra is carried out against experimental measurements.

Descriptors: Centrifugal Compressor, Large Eddy Simulation, Internal Combustion Engine, Aeroacoustics, Modal Flow Decomposition
Elias Sundström 2016, **Flödesinstabilitet i centrifugalkompressor vid lågt massflöde**
Linné Flow Centre, KTH Mekanik, SE-100 44 Stockholm, Sverige

**Sammanfattning**
Turbo spelar en viktig roll för att öka energieffektiviteten och minska föroreningar av moderna framdrivningssystem baserade på slimmade förbränningsmotorer (ICE). Centrifugalkompressorn i turboladdaren är begränsad vid icke optimala driftsförhållanden p.g.a. uppkomst av flödesinstabiliteter som orsakar roterande stall och surge. De förekommer vid reducerade varvtal (lägt massflöde), d.v.s. normala driftsförhållanden för en bättre bränsleekonomi och reducerar effektiviteten. Önskade instabila trycklaster inducerade i kompressorn reducerar även kompressorns operativa livstid. Amplifierade ljud och vibrationer kan också uppstå, vilket resulterar i betydande obehag.

Avhandlingen syftar till en fysikalisk förståelse av flödesinstabilitet och initiering av surge fenomenet med hjälp av numeriska metoder. Sådan kunskap kan möjliggöra utveckling av flödeskontroll så att turboaggregat fungerar säkrare och tystare över ett bredare arbetsområde. Bredbandig turbulensmodellering med Large Eddy Simulation (LES) har utförts och flera flödesdrivna instabiliteter har identifierats under instabila förhållanden. LES producerar stora mängder 3D-data som har efterbehandlats med hjälp av Fourier spectra och Dynamisk Modal Decomposition (DMD). Dessa metoder kan kvantifiera instabiliteter i strömningsfältet genom att extrahera stora sammanhängande flödesstrukturer och karaktärisera deras relativa andel av den totala fluktuationenergin vid tillhörande frekvenser. Bland dom mer betydande röken har en dominerande flödesmod identifierats som beskriver den fyllande och tömmande processen i samband med surge. En smalbandig struktur vid halva rotationsfrekvensen har identifierats och är relaterad till samroterande virvlar upströms om impellern samt även lokalt eleverade hastighetsmagnituder som propagerar tangentiellt i diffusorn och voluten. Dominerande flödesmoder har också hittats vid rotationsfrekvensen och högre harmoner till denna, vilka påvisar en roterande karaktär lokalisert till diffusorns inlopp.

Akustisk information och buller i kompressorn kan även erhållas från LES. Detta gör det möjligt, genom datakorrelation, att kvantifiera samband mellan flöde och akustik i kompressorn. Beräknat flöde (tryck, hastighet) och aeroakustik (ljudtrycksnivå) jämförs mot experimentella mätningar med avseende på tidsmedelvärde, fluktuerande nivåer och spektra.

**Descriptors:** Centrifugalkompressor, Large Eddy Simulation, Turboladdare, Aeroakustik, Modal Flow Decomposition
Preface

This work in fluid dynamics aims for a physics-based understanding of fluid driven instabilities developing in centrifugal compressors at unstable off-design operation. Some of the known instabilities leading to off-design operation (e.g. triggering mechanisms of rotating stall and surge) are; boundary layer separation, shear-layer instability, wake effects, vortex structures, and adverse pressure gradients. For an accurate description of these flow phenomena a broadband turbulent enabled methodology is favored such as Large Eddy Simulation (LES), which is presented. Since, LES produces a considerable quantity of 3D data, advanced modal flow decomposition techniques such as Dynamic Mode Decomposition (DMD) is utilized to extract the large coherent flow structures and characterize their relative contribution to the total fluctuation energy at associated frequencies. Thereby provide a mean to identify flow structures responsible for particular instabilities. Moreover, acoustic pressure fluctuation is extracted from the compressible LES flow solution. This enable through data correlation quantifying the flow-acoustics coupling phenomena in the compressor. Detailed comparison of flow (pressure, velocity) and aeroacoustics (sound pressure levels) predictions in terms of time-averaged, fluctuating quantities, and spectra is carried out against the experimental measurements. The Reynolds Averaged Navier-Stokes (RANS) methodology is also presented, which is orders of magnitude faster compared to LES.

In the first part of the thesis some selected fundamentals of compressible turbulent fluid flow in centrifugal compressors are introduced. A brief overview of utilized numerical methods is also presented for prediction of the compressor characteristics as well as capturing the unsteady flow field. In the second part, results from the numerical methods used are conveyed and comparison with experimental data is included from selected publications.

March 2016, Stockholm

Elias Sundström
Some people say, “How can you live without knowing?” I do not know what they mean. I always live without knowing. That is easy. How you get to know is what I want to know.

Richard P. Feynman (1918–1988)
Contents

Abstract iii

Preface v

Nomenclature ix

Chapter 1. Introduction 1
  1.1. Turbocharged Internal Combustion Engines 1
    1.1.1. The need for turbocharging 3
    1.1.2. Centrifugal Compressor & Performance Assessment 3
    1.1.3. Challenges with Turbocharged ICE 5
      1.1.3a. Limitations with low mass flow rate 7
      1.1.3b. Limitations with analytical/theoretical models 7
      1.1.3c. The need for high-fidelity simulations 7
  1.2. Thesis Objectives 8

Chapter 2. The Flow in Centrifugal Compressors 11
  2.1. The equations for compressible fluid-flow 11
  2.2. The character of the flow 12
    2.2.1. Boundary layer flow 14
    2.2.2. Boundary layer with pressure gradient 16
  2.3. Compressor Flow instabilities 16
  2.4. Compressor aeroacoustics 18
    2.4.1. Surge frequency 21

Chapter 3. Modeling Compressor Flows 23
  3.1. Challenges 23
  3.2. Turbulence modeling and challenges 24
    3.2.1. Eddy-viscosity modeling of the Reynolds stress 25
    3.2.1a. Broadband noise source models 26
Nomenclature

\begin{itemize}
\item \( c \) Speed of sound, m/s
\item \( f \) Frequency, Hz
\item \( k \) Wave number, 1/m
\item \( L \) Characteristic length, m
\item \( M \) Mach number
\item \( p \) Pressure, Pa
\item \( r, \theta, z \) Radial, Tangential and Axial coordinates, m
\item \( x, y, z \) Cartesian coordinates, m
\item \( t \) Time, s
\item \( T \) Temperature, K
\item \( \mathbf{u} \) Velocity vector, m/s
\item \( \mathbf{U} \) Mean velocity, m/s
\item \( \rho \) Density, kg/m\(^3\)
\item \( \omega \) Angular frequency, Hz
\item \( \text{RO} \) Angular velocity of the impeller shaft
\item \( \text{BPF} \) Blade passing frequency
\item \( U_{\text{redC}} \) Impeller tip speed, m/s
\item \( V \) Volume, m\(^3\)
\item \( Re \) Reynolds number
\end{itemize}

Subscript
\begin{itemize}
\item \( 0 \) Total, ambient or mean variable
\item \( \eta \) Kolmogorov scale
\end{itemize}

Superscript
\begin{itemize}
\item \( \overline{\cdot} \) Average
\item \( \sim \) Favre average
\item \( ' \) Fluctuation
\end{itemize}

Only some frequent occurring quantities and notations are listed. The other quantities are defined in the text.
CHAPTER 1

Introduction

The vehicle emission standards as directed by the European Union (EU) legislation have become more stringent over the past few decades, and further restrictions are expected to follow\textsuperscript{1,2}. The present objective dictates a 40% CO\textsubscript{2} reduction for new passenger cars and light-duty vehicles and 30% for new heavy-duty trucks by 2020. This goal is influenced by different sources. For instance, effluents from reciprocating internal combustion engines (ICE) are dangerous for humans and other living organisms, see e.g. de Alegría Mancisidora et al. (2015) for further reading. Moreover, crude oil as the main ingredient in fuel production is a thinning energy resource, see e.g. Campbell & Laherrère (1998). These are incentives for the automotive industry not only in EU but also worldwide to develop low emission vehicles with improved fuel economy.

The majority of modern automotive vehicles on the open road use a four-stroke reciprocating internal combustion engine (ICE) as the main propulsion system. Most ICE systems are based on either the Otto or the Diesel engine cycle which mainly run on oil-based fossil fuel (e.g. Gasoline or Diesel). Combustion of Diesel or Gasoline emits species that are dangerous for humans and other organisms, e.g. carbon monoxide (CO), nitrogen oxides (NO\textsubscript{x}), hydrocarbons (HC), particulate matter (PM), or contribute to global warming mainly (CO\textsubscript{2}). Reduction of fuel consumption and CO\textsubscript{2} emissions is thus not only of direct economic interest for the vehicle operator but also for the environment. Emission of CO, NO\textsubscript{x}, HC, and PM are currently limited by means of catalytic converters and/or particle filter systems attached to the exhaust pipe system, see works by Bhattacharjee et al. (2011); Needham et al. (2012); Koltsakis et al. (2009).

1.1. Turbocharged Internal Combustion Engines

The principle of the ICE is injection of air and fuel into the cylinders, compress the air and fuel mixture, which is then ignited with a spark plug or self-ignited


\textsuperscript{2}\textsuperscript{http://ec.europa.eu/clima/policies/transport/vehicles/heavy/documentation\_en.htm}
due to the compression rate. The chemical energy content in the mixture is released as heat. The heat release of hot gases results in elevated thermal energy levels, which is subsequently expanded and converted into kinetic energy which drives the piston. The piston work is then transferred into angular momentum of the connected crank shaft. Through the power-train system the angular momentum is transferred into usable wheel torque which propels the vehicle forward. The engine power output can be approximated with the following equation:

\[ P = \frac{1}{2} \cdot n \cdot p_{me} \cdot V_{SW} \]  

(1.1)

This elementary equation shows that an increment of either the engine speed \( n \), the brake mean effective pressure \( p_{me} \) (a measure of the engine load) or the swept cylinder volume \( V_{SW} \) (i.e. engine size) results in an increase of the engine power. However, they result in different engine friction losses. With a first-order estimate, the fiction losses: increase like the square of the engine speed, is approximately constant with the engine load, and increase linearly with the engine size, see works by Guzzella & Sciarretta (2007); Ben-Chaim et al. (2013); Leduc et al. (2003).

A smooth operation with minimum noise and vibration levels defines a lower engine speed bound. Hence, reduced engine size (e.g. downsizing) is a more promising course for augmented engine efficiency. For a given power output, downsizing is counteracted with an increased engine load. Figure 1.1 shows a specific fuel consumption map for a Diesel engine with displacement of 2 liters. The map is adapted from the works by Guzzella & Sciarretta (2007); Ben-Chaim et al. (2013); Leduc et al. (2003).

![Specific fuel consumption map](image)

Figure 1.1: Engine performance map, adapted from Guzzella & Sciarretta (2007); Ben-Chaim et al. (2013); Leduc et al. (2003).

(2007); Ben-Chaim et al. (2013); Leduc et al. (2003) and illustrate more fuel
efficient operation at low engine speed and high engine load for the same engine power output. Downsizing also reduces the overall weight of the engine system, which eases demands for high power output in the event of vehicle acceleration operation.

1.1.1. The need for turbocharging

Increased engine load demand elevated fuel burn rates, but this requires increased air supply to the cylinders to ensure an optimal air-to-fuel ratio. This can be obtained using a turbocharger system which exercise growing attention for improved energetic efficiency and reducing pollutants of the modern power-train systems based on downsized ICE. The turbocharger consists of a turbine and a compressor, see Fig. 1.2 and Fig. 1.3. It is connected through gas exchange piping with the engine. The turbine is spooled to high rotational speeds by harnessing and converting some of the exhaust gas enthalpy into kinetic energy. A centrifugal compressor, also known as radial compressor and is classified as dynamic compressor, is mounted on the same shaft as the turbine. Figure 1.3 shows a turbocharger hardware assembly with the centrifugal compressor system located to the left, a housing/hub rotating assembly in the middle also known as an oil housing system, and a turbine system to the right. The housing of the individual assembly subsystems has been sliced to expose internal parts inside such as the impeller, diffuser, and shroud. The particular compressor system depicted in Fig. 1.3 is equipped with a bypass channel, which serves as a surge controlling device. The main function of the centrifugal compressor system is to provide a pressure rise by a dynamic transfer of kinetic energy, which increase the velocity of a continuous flowing fluid stream through the impeller or rotor. This kinetic energy is converted to a potential energy which results in an increased static pressure by compressing the flow through a diffuser to low velocities. The higher static pressure results in an increased air density, which is supplied at the engine intake manifold, thus forcing more air into the cylinder. Feeding the engine with a higher density compressed air, reaching its oxygen demand is key for a better combustion. A charge air cooler is beneficially mounted after the compressor to further elevate the intake air density.

1.1.2. Centrifugal Compressor & Performance Assessment

Under design operating conditions with approximately steady-state flowing fluid stream this pressure rise is obtained efficiently with limited losses in the energy transfer. However, the operation of the centrifugal compressor deteriorates at off-design operating conditions by the inception of flow instabilities causing rotating stall and surge, see works by Jansen (1964); Greitzer (1976); Fink et al. (1992); Tsujimoto et al. (1996); Guillou et al. (2012). They occur at reduced engine speeds (low mass flow rates), i.e. desired operating conditions for optimum fuel economy. The surge flow phenomena occur when the
static pressure at the compressor outlet is so exalted that the flow reverses in the impeller and is forced upstream through the compressor inlet. This limit bound is indicated with a surge line in a centrifugal compressor map as shown in Fig 1.4. The map is drawn schematically and illustrates pressure rise and efficiency contours as function the mass flow at different constant rotational
speed lines. Desired operation is at high efficiency along the design line. Since a centrifugal compressor can be designed in many different ways, (e.g. different number of blades, with or without splitter blades, vaned or vaneless diffuser, ported shroud or active surge control, backward or forward swept trailing edges etc.) the location of the surge line is typically unique for the considered compressor. Commonly the performance map is obtained from measurement on a gas stand under idealized installation. When the compressor is integrated as part of an ICE system, the surge line may shift due to upstream and downstream installation effects. According to the works by Greitzer (1976) surge normally occurs at zero or positive slope of the performance curve but is usually computed as the lowest mass flow rate possible to record a stable reading from the gas stand. From an engineering point of view an additional surge margin is imposed on the actual compressor surge line. This is an extra margin to avoid surge, and is the difference between the actual surge line and the surge control line used by the surge control system. It does not indicate an ultimate transition point from stable to unstable operation but is more considered as a sufficient safety margin from where the compressor is likely to go into unstable operating regimes. There are a number of different definitions of the surge margin in use, commonly a few percent offset to the actual surge line, see works by Bloch (2006). For this reason a universal definition of the actual surge line and the safety margin for any compressor design is challenging to define.

1.1.3. Challenges with Turbocharged ICE

One consequence with the surge phenomena, resulting in the large amplitude pressure fluctuations in the compressor were analyzed by Leduc et al. (2003).
They investigated the engine torque as function of the engine speed for: a naturally aspirating engine, a downsized engine with conventional turbocharger, and a downsized engine without turbocharging, respectively. The study concludes possible engine capacity reduction by 60% with maintained torque under near optimum design operating conditions. However, issues with low end torque as with starting and transient torque with downsized engines are highlighted. The second issue with downsizing and turbocharging is that the pressure ratio increases in the combustion cylinders. Such conditions may introduce issues with knock and high exhaust temperatures which may lead to structural damage and fatigue of the engine system. A third issue from the vehicle operator’s point of view is the acceptance of having low end torque as well as the cost associated with adding a turbocharger. Amplified noise is also generated during the surge operation, resulting in a notable discomfort for the driver and passengers of the road vehicle. The lack of knowledge associated with the compressor flow instabilities occurring at these operating conditions reduces significantly the possibility of improving the performance of turbocharger power-train systems.

To assess any gain in fuel economy with turbocharged downsized engines it is relevant to evaluate how often the turbocharger system operates near the surge condition, i.e. at non-optimal conditions. The European Union utilizes a theoretically derived drive cycle called New European Driving Cycle (NEDC) for passenger cars, see e.g. Pacheco et al. (2013). NEDC dictates the vehicle speed as function of time, where emission standards must be fulfilled. Approximately two-thirds of the cycle is subjected to low speed urban driving with an average speed in the order of 20 km/h whereas the remaining part of the cycle is subjected to highway driving with an average speed in the order of 80 km/h. This proposes that the engine, in most of the drive cycle would operate at low end torque and or starting and transient torque. Under such conditions with low engine speed the amount of exergy is limited and may not be sufficient to spool up the turbocharger and start producing boost pressure. The operating speed at which there is enough exhaust gas enthalpy and hence possible to force compressed air into the cylinders is called the “boost threshold rpm”. The boost threshold may depend on several factors but primarily on the engine displacement, engine rpm, throttle opening, and the size of the turbocharger. Therefore, the boost threshold needs to be obtained either experimentally or by means of system simulation. Once the boost threshold rpm is known for a particular turbocharger design, a system based simulation may be performed that uses the NEDC to obtain a quantitative fuel economy for the vehicle system.

To ensure safe and stable operation of the turbocharger and preventing long durations of large amplitude pressure pulsations which can be damaging for the compressor, surge control is commonly employed. There exist passive and active surge control techniques with the main function to lower the peak pressure rise and remedy flow oscillations. With active surge control some
of the compressed air is ventilated through a bypass valve. When activated, additional boost pressure would be wasted and the potential ICE efficiency increase would default. Passive surge controls may consist of a ported shroud cavity with open ports in close proximity of the impeller blade tips in the region of the shroud where back-flow is expected at off-design operating conditions, see Fig. 1.5. The reversed flow is subsequently ventilated through the open ports into the ported shroud cavity where it is guided away from the impeller eye. Some distance upstream this flow is directed back into the air-intake of the compressor. The overall effect achieved is that the surge line and hence surge margin is shifted to lower mass flow rates in the compressor map which hence widens the operating range at the expense of reduced maximum pressure rise.

1.1.3a. Limitations with low mass flow rate. From the engine efficiency and early integration of turbocharging ICE systems perspective its necessary with further improvements. For instance, optimizing performance also under unsteady off-design operating conditions may be beneficial, since it can occur during realistic and desired engine working situations. Of particular importance is accurate predictions and knowledge of inception of the flow driven instabilities causing rotating stall and surge which leads to unstable operation, which harms the ICE efficiency and demands provisions for surge controlling.

1.1.3b. Limitations with analytical/theoretical models. Predictive computational models are essential for ICE manufacturers. Intensively used in the automotive industry, e.g. in early compressor design studies or to best integrate an ICE with a turbocharger. Such models are simplistic and robust so that they can provide design parametric studies in affordable time-frames, see works by Gravdahl & Egeland (2012); Greitzer (1976); Fink et al. (1992). They are 0D/1D models typically based on steady-state assumptions instead of considering unsteady three-dimensionality of the flow in typically complex geometries. These are reasons for why the models fail predicting accurately the performance parameters of a compressor when approaching the unstable operating conditions occurring during realistic driving conditions. However, an improved predictive performance can be achieved by calibrating and validating the models against reliable experimental data measurements.

1.1.3c. The need for high-fidelity simulations. Compressible Large Eddy Simulation (LES) calculations allow capturing of the fluid-flow physics associated with the compressor operating under unstable conditions. Figure 1.5 illustrates the typical chaotic flow field emerging under unstable surge operation as compared with near optimum design condition for a ported shroud centrifugal compressor. The streamlines are presenting qualitatively the differences between the two flow scenarios, with severe flow disturbances and induced secondary-flow motion observed at off-design operating conditions in the impeller eye’s
region, diffuser and compressor’s volute. The purpose with the ported shroud

![Flow field visualization](image)

Figure 1.5: Instantaneous visualization of the flow field in a ported shroud centrifugal compressor under unstable surge operation as compared with near optimum design condition.

is to allow some flow to recirculate back from the impeller to the compressor inlet. This is known to widen the operating range near the surge-line, see e.g. Guillou et al. (2012). For an accurate representation of the interaction between the incoming flow and the impeller, the computational grid associated with the impeller is rotating at the prescribed RPM, according to the compressor’s operating condition. The chaotic flow field contained in the LES data requires advanced post-processing techniques, i.e. Fourier spectra, Proper Orthogonal Decomposition (POD) and Dynamic Mode Decomposition (DMD). These techniques are capable of calculating the instabilities in the flow field by extracting large coherent flow structures and characterize their relative contribution to the total fluctuation energy at associated frequencies. Thus, one can identify the flow structures responsible for particular instabilities, which allows developing mitigation techniques of these unwanted phenomena (e.g. by using flow control). Moreover, from the compressible LES flow solution one can extract acoustic information quantify flow-acoustics coupling phenomena in the compressor. Predictions of flow (pressure, velocity) and aeroacoustics (sound pressure levels) in terms of time-averaged, fluctuating quantities, and spectra are performed to assess narrowband and broadband features in the flow field.

1.2. Thesis Objectives

The long term goal for this thesis is to enhance the understanding of compressor flow dynamics associated with off-design operating conditions, promoting a physics-based understanding of the compressor flow instabilities and surge inception phenomenon. The triggering mechanism has been hypothesised to be
linked with flow driven instabilities such as adverse pressure gradients, shear-layer instabilities, boundary layer separation and wake effects. Due to the confined space, complex geometry, rotating impeller, and compressible flow with wide range of temporal and spatial length scales, it is challenging from an experimental point of view to pinpoint the triggering mechanism. For this reason, the LES approach is employed for capturing flow driven instabilities that might be present in off-design condition. The flow instabilities and their evolution in space and time can be exposed using flow mode decomposition techniques. Thus the evolution of the instabilities leading to surge can be followed when moving from design operating conditions to off-design conditions occurring with low mass flow rates. From the acoustic point of view the hypothesis is that there is a flow-acoustic coupling that makes centrifugal compressors go into rotating stall. There is also the hypothesis that a certain mechanism causes amplified noise levels such as tip leaking noise. For instance when a blade tip vortex is disrupted, noise might be emitted and therefore, one should see the imprint in the flow mode shape or the Fourier surface spectra.

Computing the flow mode shapes may answer if surge emerges from a flow-acoustic coupling or not. First, from DMD modes based on pressure in the compressor including the exit pipe, one can check if the pressure information travels with the speed of sound. Secondly, there should be a peak that travels with the speed of sound at the surge frequency towards the compressor. For sure there will be acoustic waves, which are reflected at the outlet. However, those might not have a wavelength corresponding to the surge frequency. If one estimates the wavelength corresponding to a surge frequency at say 43 Hz, one obtains something in the order of 4-5 meters. This is rather long compared to compressors used in road vehicles. Such an acoustic wave would need to be emitted somewhere at the impeller, since it cannot be generated at the outlet nor at the inlet. However, the wavelength is rather large compared any dimensions of the impeller. Therefore, it is conjectured that one should see a standing wave and not a wave peak traveling back into the compressor. This would state that the wave is a compression wave and not an acoustic wave. With a compression wave, the exit pipe is "filled" to a very high pressure when the impeller is very efficient and it is "emptied" to a lower pressure when the impeller is inefficient. One may verify this by looking at the pressure gradient of the DMD mode in the exit pipe’s axial direction.

By evaluating the DMD mode based on velocity at for instance 50% of the impeller’s blade span, one may see repetitive occurrence of flow separation with the surge frequency. Thereby, one may explain the pulsation of the global flow. However, one should keep in mind that surge is a global and not a local phenomenon. Boundary layer separation is local and that is why it doesn’t necessarily link to surge. Roughly one can explain one surge cycle as pressure building up from the exit and propagating towards the impeller (filling). There at the impeller, some flow is pushed as tip leakage over the impeller blades.
This recirculating flow causes the swirling flow at the impeller eye, which alters the incidence angle of the flow entrained into the impeller. Thereby the efficiency of the impeller is degenerated. The reduction of momentum transferred downstream by the impeller results in a pressure decrease in the exit pipe. Because the downstream pressure decreases, the amount of tip leakage decreases and further the incidence angles change back. The impeller becomes more efficient again and the surge cycle starts from the beginning. This is a very short description of surge without details. The point is that boundary layer separation may be involved.

If the operating condition for the LES calculation is chosen in surge, one can describe how it manifests. But none of this explains why it is there or why it occurs at different mass flow rates. In the compressor map, assuming steady conditions, the inlet conditions do not change. And on one speed line of the compressor, the rotating speed does not change. The only thing that changes is the outlet boundary condition. Hence, the imparted work of the impeller is essentially the flow momentum transferred downstream, which is reduced by the pressure gradient against the flow has to push. Also there are some losses, as e.g. turbulence dissipation, recirculating flow, which we call just losses for now. However, the essential parameters to characterize the work done by the impeller are the flow momentum transferred downstream and the pressure gradient. (Both are not influenced by the assumed constant inlet boundary conditions.) Along one speed line, the downstream boundary conditions govern the pressure gradient by restricting the mass flow rate. The more the mass flow is restricted at the outlet, the higher the pressure gradient will be in general (neglecting losses). This leads to a reduction of imparted work into the system. I) Work is only extracted through the outlet and dissipated by the losses. The amount of work extracted is limited by the outlet boundary condition and it decreases with reduced mass flow rate. II) As long as the work that is imparted can be extracted through the outlet, the global system will be stable (in the sense that it doesn’t surge). All flow momentum fluctuations will be just swapped out at the outlet. If the outlet doesn’t allow the imparted compression work to leave, it will dam up as compressed flow in the outlet pipe, and the surging starts. Hence, we need to estimate the slope (as a function of mass flow rate) of the imparted work, the losses and the work allowed to leave the compressor. The intersection of this line is the point where the surging starts. It is being called work here, but essentially one can see it as an energy balance.
CHAPTER 2

The Flow in Centrifugal Compressors

2.1. The equations for compressible fluid-flow

The air in the compressor is a compressible fluid and governed by the mass, momentum and energy conservation equations, respectively, for a derivation of these equations see e.g. works by Hoffmann et al. (1996). In tensor notation they are:

\[ \frac{\partial \rho}{\partial t} + \frac{\partial \left( \rho u_i \right)}{\partial x_i} = 0 \]  (2.2)

\[ \frac{\partial \left( \rho u_i \right)}{\partial t} + \frac{\partial \left( \rho u_i u_j \right)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} + \rho f_i \]  (2.3)

\[ \frac{\partial \left( \rho e_0 \right)}{\partial t} + \frac{\partial \left( \rho u_i e_0 \right)}{\partial x_i} = -\frac{\partial \left( p u_i \right)}{\partial x_i} - \frac{\partial q_i}{\partial x_i} + \frac{\partial \left( u_j \sigma_{ij} \right)}{\partial x_i} \]  (2.4)

where \( t \) – time, \( \rho \) – density, \( x_i \) – Cartesian coordinate (\( i = 1, 2, 3 \)), \( u_i \) – absolute fluid velocity component in direction \( x_i \), \( p \) – static pressure, \( \sigma_{ij} \) – stress tensor components, \( q_j \) – heat flux, \( f_i \) – external body source components, and \( e_0 = e + \frac{1}{2} u_i u_j \) is the total energy per unit volume.

The compressed fluid is also assumed to be Newtonian and obey the following constitutive relation connecting the components of the stress tensor to the velocity gradients:

\[ \sigma_{ij} = 2\mu S_{ij} - \frac{2}{3} \mu S_{kk} \delta_{ij} \]  (2.5)

where \( \mu \) is the molecular dynamic viscosity of the fluid and \( \delta_{ij} \) is the Kronecker delta function. The rate of strain tensor is given by:

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  (2.6)

The viscosity dictates how the fluid resist deformation due to shear stress, (e.g. where neighboring fluid particles slide past each other at different speeds). The effect of the shear stress is generation of a force opposite to the flow direction. Since the temperature of the compressed fluid increases, the dynamic viscosity is no longer independent of the temperature. This dependence is obtained from
the Sutherland equation:

$$\mu = \mu_{ref} \frac{T_{ref}}{T + T_{suth}} \left( \frac{T}{T_{ref}} \right)^{3/2}$$  \hspace{1cm} (2.7)

where $\mu_{ref}$ is the dynamic viscosity at $T_{ref} = 273.15$ K and $T_{suth} = 110.4$ K is the Sutherland constant. It is assumed that molecular diffusion fluxes of heat and mass satisfy Fourier’s law:

$$q_j = -k \frac{\partial T}{\partial x_j}$$  \hspace{1cm} (2.8)

where $k$ is the thermal conductivity. This is also obtained from Sutherland’s equation, with reference conductivity $k_{ref} = 0.02614$ W/m-K.

Equations (2.2) to (2.4) is a non-linear system with more unknowns than equations. It is closed using the ideal gas law and provides a link between the momentum equations and the energy equation:

$$p = \rho RT$$  \hspace{1cm} (2.9)

where $R$ is the gas constant, which is the relation for specific heat capacity at constant pressure $c_p$ and constant volume $c_v$, respectively, and $T$ is the temperature. Thus, assuming that the fluid media behaves like a perfect gas and where intermolecular forces can be neglected. Assuming that the fluid is a thermally perfect gas the internal specific energy $e$ and the specific enthalpy $h$ are only dependent on temperature. Moreover, since the fluid only reach moderate temperatures (e.g. $T < 1000$ K), the fluid can also be assumed to be calorically perfect where specific heats are constants. Thus, the specific internal energy and the specific enthalpy are given as follows:

$$e = c_v T$$  \hspace{1cm} (2.10)

$$h = c_p T$$  \hspace{1cm} (2.11)

Due to the impeller rotation the fluid is subjected to rotational forces. This can be introduced as a momentum source field in Eq. (2.3) and is given by:

$$f_i = f(u_i, \omega_i, r_i)$$  \hspace{1cm} (2.12)

where $f()$ is a function of the rotation vector $\omega_k$ and radius vector $r_k$ about the impeller axis.

### 2.2. The character of the flow

The flow field in the centrifugal compressor is by large subjected to a highly irregular and rotational flow motion. This is also known as a turbulent flow where inertial forces dominate over viscous forces. The turbulent regime is commonly specified using the non-dimensional Reynolds number, which may also be viewed as the fraction of turbulent diffusion over molecular diffusion. If a centrifugal compressor with a characteristic length scale $L \approx 0.1$ m is
2.2. THE CHARACTER OF THE FLOW

The time scale for the molecular diffusion process can be estimated by:

\[ T_{mol} \sim \frac{L^2}{\nu} \approx 11 \text{ min} \]  \hspace{1cm} (2.13)

where \( \nu = 1.5 \cdot 10^{-5} \text{ m}^2/\text{s} \) is the kinetic viscosity of the fluid. In contrast, the time scale for a turbulent diffusion process is typically orders of magnitude faster than the time scale associated with molecular diffusion and it’s also governed by a large range of scales of the eddy sizes. The time scale for this process can be estimated as:

\[ T_{turb} \sim \frac{L^2}{\nu_{turb}} \approx \frac{L^2}{A \cdot u'} \approx 5 \text{ millisec} \]  \hspace{1cm} (2.14)

where \( A \) is a typical eddy size of say 50 percent of \( L \), and with a characteristic fluctuating velocity \( u' \) of say 10 percent of the impeller tip speed. The \( Re \) is subsequently computed as:

\[ Re_A = \frac{\text{inertial forces}}{\text{viscous forces}} = \frac{\Lambda u'}{\nu} > 10^5 \]  \hspace{1cm} (2.15)

Some of the typical features of the centrifugal compressor flow are hence: high diffusivity, high Reynolds number and thus subjected to a three-dimensional chaotic nature. The largest eddy scales are limited by the geometry (e.g. the characteristic diameter and the boundary layer thickness) and these receive energy input from the mean flow. It is generally accepted that a cascade process takes place where energy is transferred from the interaction of large eddy scales to successively smaller eddy scales. Ultimately, molecular viscosity defines a lower limit of the eddy size which is then dissipating its remaining kinetic energy into heat. The cascade mechanism is related to vortex stretching which thus requires treatment of a three-dimensional flow. In the works by Kolmogorov it was hypothesized that the cascade process leads to a self-similar state of small eddy sizes, whereby small scales are independent of large scales. In his theory it is assumed that the smallest length scale \( \eta \), time scale \( t_\eta \) and velocity scale \( v_\eta \) depends only on the dissipation of kinetic energy per unit mass and time, dissipation rate \( \varepsilon \) and the kinetic viscosity. With dimensional analysis the following relations are obtained:

\[ \eta = \left( \frac{\nu^3}{\varepsilon} \right)^{1/4}, t_\eta = \left( \frac{\nu}{\varepsilon} \right)^{1/2}, v_\eta = (\nu \varepsilon)^{1/4} \]  \hspace{1cm} (2.16)

\[ Re_\eta = \frac{\eta \cdot v_\eta}{\nu} = 1 \]  \hspace{1cm} (2.17)

Assuming a power input in the order of 10 kW and a total air mass in the order of 60 g gives \( \varepsilon \approx 2 \cdot 10^5 \text{m}^2/\text{s}^3 \) and subsequently \( \eta \approx 10 \mu \text{m} \). One consideration for high \( Re \) turbulent flow is the quasi equilibrium assumption. It dictates that energy flux from large eddy scales to small eddy scales are in equilibrium.
2. THE FLOW IN CENTRIFUGAL COMPRESSORS

\( \epsilon_f = \epsilon \). The ratio of largest to smallest scales can be shown to be given by:

\[
\frac{\Lambda}{\eta} = \left( \frac{u \Lambda}{\nu} \right)^{3/4} = Re^{3/4}_{\Lambda}
\]

which implies that the number of degrees of freedom for a 3D flow scales as \( Re^{9/4}_{\Lambda} \). For turbulence modeling of the cascade process of all eddy scales then means that the number of grid points grows as \( Re^{9/4}_{\Lambda} \) and with a computational effort growing as \( Re^3 \). With the previous estimate of the Reynolds number it suggest a need for \( 3 \cdot 10^9 \) grid points, which is a large number and challenging to realize on available hardware resources.

2.2.1. Boundary layer flow

There are different flow regimes taking place near the wall in the so called thin boundary layer. In laminar flows it is observed that the velocity increases approximately linearly in the wall normal direction. In fully turbulent regimes this is observed to change more according to a logarithmic increase. There may also be situations with a transition from laminar to a fully developed turbulent boundary layer. With the high Reynolds number flow over for example impeller blades there may be a mix of laminar and turbulent flow regimes in the boundary layer. Closest to the non-slip wall there is a laminar sublayer which requires computational effort to resolve with the grid. An approximate tangential velocity variation with respect to the wall normal direction is obtained using the thin shear-layer equation:

\[
U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = \frac{-1}{\rho} \frac{dP_0}{dx} + \frac{\partial}{\partial y} \left( \nu \frac{\partial U}{\partial y} - u' v' \right)
\]

This equation results from the continuity and Reynolds equations assuming: 2D flow, \( Re \gg 1 \), and steady flow. It can be seen that the validity range is limited to boundary layer flows, channel & pipe flows, wake flows, jet flows and mixing layer flows. Thus, care must be taken to adopt it for unsteady flows in centrifugal compressors. Introducing non-dimensional quantities, \( y^+ = \frac{yu}{\nu} \), \( u^+ = \frac{u}{u_\tau} \), where \( u_\tau = \sqrt{\tau_w/\rho} \) is the friction velocity, and \( \tau_w \) – wall shear stress. Inserted into Eq. 2.19 together with Eddy-viscosity based modeling of the turbulent stress and assuming that the velocity scales as \( \sqrt{K} \) and that the mixing length scale varies linearly from the wall, the following relation is obtained:

\[
u^+ = \frac{1}{\kappa} \ln (y^+) + B
\]

where \( \kappa \) – von Karman constant, and \( B \) – empirical coefficient. This is the so called log-law which is approximately valid sufficiently far from the immediate surface in a region where \( 50 \leq y^+ \leq 200 \). For the inner most layer \( y^+ \leq 5 \) the cross stream profile approximates to \( u^+ = y^+ \) which governs the viscous sublayer. Outside the log-law region there is a wake-region at wall normal...
2.2. THE CHARACTER OF THE FLOW

Distances exceeding approximately 15% of the boundary layer thickness, see Fig. 2.1. The blended wall law from the works by Reichardt (1951) is intended to represent the buffer layer by combining the viscous sublayer and the logarithmic region and is used for all $y+$ wall treatments. Using wall functions ease the computational effort by allowing coarser grids near the wall. However, it only applies to attached flow and unfortunately fails in recirculating and separated flow (e.g. under surge operations). Close to the leading edge of the blade surface the boundary layer flow is typically subjected to a laminar flow unless it is tripped into turbulence with a surface roughness device. This is illustrated with the thick black line in Fig. 2.1 at 40% blade chord.

As the fluid particles travels over the non-slip wall, surface roughness will gradually disturb the laminar flow. For long enough distances a laminar-to-turbulent transition might take place. Since, the blade chord in centrifugal compressors is relatively short (in the order of a few cm) as compared with for example aircraft wings (in the order of meters), then a significant part of the boundary layer may be laminar. Only a small part towards the trailing edge might have a gradually log-law characteristic layer, as indicated with the thick green curve at 90% chord. Nevertheless, its also important to note that the boundary layer state, i.e. laminar or turbulent or a mix thereof, over the impeller surface also depends on upstream turbulence levels. For instance a very long upstream pipe installation may have a completely different characteristic compared to a relatively short idealized bell mouth installation. The integral length scale is commonly used to assess the boundary layer thickness and can

Figure 2.1: Law of the wall, tangential velocity near the wall with mixing length model as compared with LES data at the impeller blade surface under approximate steady-state conditions
be computed with the displacement thickness as follows:

\[ \delta_1 = \int_0^s \left(1 - \frac{\rho u}{\rho_e U_e}\right) dy \]  
(2.21)

Assuming a compressible laminar boundary layer then this is approximately 25% of the blade-to-blade pitch corresponds to \( y^+ \approx 500 \) at 40% chord and \( y^+ \approx 250 \) at 90% chord in Fig. 2.1. The momentum loss thickness is computed as:

\[ \delta_2 = \int_0^s \frac{\rho u}{\rho_e U_e} (1 - u/U_e) dy \]  
(2.22)

which is approximately equal 8% of the blade-to-blade pitch. If the flow is attached (e.g. stable design operating conditions) the blended law is a viable option and is computationally an affordable option.

2.2.2. Boundary layer with pressure gradient

In some situations the boundary layer might be subjected to an adverse pressure gradient (APG). From Eq. 2.19, there is an APG when \( \frac{dp}{dx} > 0 \). This causes a deceleration of the boundary layer flow and introduces turbulence production. In particular there is a positive contribution emerging from diagonal stress terms in the kinetic energy equation, see derivation in Pope (2000). For strong enough APG, the fluid in the boundary layer may slow to zero velocity or even become reversed. In case of flow reversal, the flow is said to be separated from the surface. In the case of a centrifugal compressor the blades are indeed subjected to a strong adverse pressure gradient where the pressure is gradually increasing in the blade passage towards the exducer. This means that there is in general always a potential risk for boundary layer separation over the blades.

2.3. Compressor Flow instabilities

Boundary layer separation on the blade surfaces is one possible route to compressor flow instabilities, i.e. a possible triggering mechanism for rotating stall and surge. If it happens the flow is reversed in the boundary layer and a pressure equalization takes place where the pressure increases in the blade passage. A high enough pressure in the blade passage implies that the flow is partially blocked and the blades can no longer produce sufficient momentum and push against the high pressure downstream in the diffuser passage and the overall pressure ratio drops. Different compressor types have different stalling characteristics and it is thus an important phenomenon that affects the performance of the compressor. A positive stall separation is said to occur on the suction side of the blade. A negative stall separation is said to occur on the pressure side of the blade. In general, negative stall is negligible as compared with positive stall because it is more likely that separation will occur on the suction side of the blade. If the flow separation is more or less uniform on all blades then this is said to be a so called stationary stall. However, the air-flow in
centrifugal compressors is in most situations non-uniform over time due to the rotation of the impeller blades disturbing the local incoming air flow at the impeller eye. If the compressor is fitted with an asymmetric volute, an adverse pressure gradient may develop right under the volute tongue. This may also lead to nonuniform flow in the diffuser. During rotation some blades may have a higher incidence or angle of attack to the flow compared to neighboring blades. Say if the impeller is rotating clock wise then the left blade will be subjected to a higher incidence and be subjected to stall whereas the right neighboring blade has a lower incidence and experience lesser stall. The higher pressure in the blocked passage in between the blades causes a disturbance of the flow and causes an increases incidence on the right blade so that the right blade will start to experience more stall. This process is gradually shifted from blade to blade manifesting in a so called rotating stall. In the event of rotating stall, a precursor to the surge phenomena, there can thus be one or more stalled passages also known as stalled cells. The propagation under which the stalled cells move between blade passages is typically in the order of 50% of the rotating order. The exact mechanism for triggering rotating stall in centrifugal compressors is not completely understood but for axial compressors there are at least two known distinct routes with different onset criteria for each, see works by Camp & Day (1997); Pullan et al. (2015). In these works the inception mechanism of rotating stall is discussed based mainly on support from experimental studies for a low-speed axial compressor as well as accompanying RANS calculations on a sector mesh. The discussed routes are: (i) modal stall (i.e. small growth of disturbances with length scale of the compressor circumference), (ii) spike stall (related to a blocked blade passage with length scale one or several blades). Some observation of the so called “spike stall” is that the length scale of disturbances is in the order of one/two blade passages. The disturbances cause an increased pressure in the blade passage, and thus relates with flow blockage in the passage. When this happens above references argues that it triggers a boundary layer separation on the suction side near the outer part of the blade leading edge, which rolls up into a vortex. This vortex is being convected downstream and up to the shroud wall. During the vortex formation the pressure drops on the blade suction side. The high pressure is followed by a low pressure. This sequence is gradually shifted to the next neighboring blade passage and repeated. Overall, this event is coined “spike stall”. However, this is not necessarily a general mechanism for onset of rotating stall in centrifugal compressors. An alternative route to rotating stall for high-speed centrifugal compressors with vaned diffuser is discusses in the works by Tréhinjac et al. (2011). Based on RANS computations validated with experimental data the onset of rotating stall is traced to boundary layer separation on the vane pressure side being transferred to the suction side. It was found that pressure waves cause an alternation of favorable and adverse pressure gradients which leads to periodic separation and reattachment of the
boundary layer in the diffuser passage. Moreover, a severe hub/suction side corner separation is argued to be the origin of surge inception.

If the compressor is demanded to operate at even lower mass flow rates, a large system pressure oscillation is developing with complete flow reversal in the impeller. The surge frequency is typically in the order of dozens of hertz or approximately a few percent of the rotating order. This is known as the unwanted surge phenomena which can be damaging for the compressor structure. There exist various ideas for the triggering mechanism causing surge. One possible route is the hypothesis that a strong shear-layer in the volute associated with a velocity gradient is part of the surge instability, see works by Guillou et al. (2012). The relative strength of the shear-layer is said to be oscillating corresponding to the surge frequency and believed to origin under the volute tongue.

Ported shroud centrifugal compressors are known to induce so called secondary flow instabilities at near surge conditions. In the works by Guillou et al. (2012); Jyothishkumar et al. (2013); Semlitsch et al. (2013), the interaction of the reversed flow from the ported shroud cavities with the incoming air flow was studied using Particle Image Velocimetry (PIV) measurements as well as supported with LES calculations. Unsteady shear-layers were detected as well as the development of four vortex structures. These were seen to be associated with the flow reversal off the port cavities observed under surge conditions. In follow up works by the thesis author Sundström et al. (2014, 2015a) co-rotating vortices were located upstream of the impeller face starting at the non-slip walls at the bell mouth entrance. Distinct mode shapes were found in Fourier surface spectra as well as in the DMD analysis at the rotating stall mode frequency. The co-rotating vortices were seen to be stretched and eventually breaking up into smaller eddy structures before entering the impeller eye.

### 2.4. Compressor aeroacoustics

Aeroacoustics is a discipline in fluid dynamics that investigates the aerodynamic generation of sound, its propagation and its effects. The aerodynamically generated sound can be seen as a disturbance in pressure that propagates through a fluid with an acoustic velocity. In air this is the speed of sound. Naturally, the disturbances are emitted from certain sources and are associated with fluctuations overlapping the atmospheric pressure. The sources can be generated by free fluid motion (e.g. turbulent shear-layers) or by the fluid flow interacting with solid structures (e.g. rotating impellers). The meaning of noise can be defined as unwanted or damaging sound. The Sound Pressure Level (SPL) in dB scale is a measure of perceived sound, starting at 0 dB for human beings. Exposure to compressor surge noise with levels exceeding 130 dB (e.g. threshold of pain) can therefore be damaging and can cause permanent hearing loss.
The SPL is related to energy carried by sound waves and is computed as:

\[ SPL(dB) = 20 \log_{10} \frac{p_{rms}}{p_{ref}} \]  

(2.23)

where \( p_{rms} \) is the root-mean-square value of the pressure fluctuation and \( p_{ref} = 2.0 \cdot 10^{-5} \) Pa is the reference sound pressure (e.g. threshold of human hearing).

Lighthill (1952) derived a theory for predicting the sound intensity radiating in turbulent flows. The frequency of a tone is related to the Strouhal number, which is a non-dimensional frequency based on result from quantitative observations. The theoretical work provides a concept of aeroacoustic analogy which replaces the actual flowfield responsible for generating noise with an approximate system of noise sources. The noise sources act on a uniform steady fluid in the framework of acoustic propagation equations. An accurate description of the sources becomes therefore an issue in noise predictions. The theory was extended in the works by Curle (1955) to include the effect of flow-body interaction on sound generation. Another extension is from Williams & Hawkings (1969) who takes into account arbitrary surface motion, resulting in a formulation targeting noise prediction in rotor blade aerodynamics (e.g. turbomachinery).

One of the main mechanisms of sound generation in the presence of solid structures is vortex shedding noise where vorticity is released from a bluff body. The fluctuating forces on the body in the unsteady flow are transferred to the fluid and propagate as sound. Turbulent eddy structures impinging on a solid surface generate local pressure fluctuations on the body surface which feed the acoustic farfield. Trailing edge noise is important for centrifugal compressors, due to interaction of boundary layer instabilities with impeller tip edges.

The acoustic noise generated by centrifugal turbomachinery of Diesel engines has become a principal concern for the construction design due to the notable discomfort induced on the passengers, see Broatch et al. (2015); da Silva Brizon & Medeiros (2012). The acoustic noise generation on the compressor side of the turbocharger becomes a challenge, especially during compressor off-design operating conditions at low mass flow rates, see Wenzel (2006). Under such circumstances, the engine noise is inherently diminished, and does not mask the turbocharger noise at such operating conditions, which becomes therefore distinctively audible, as studied by Gonzalez et al. (2003). In the recent years, with the adopted trends concerning engine downsizing, the provoked noise by the compressor may contribute substantially to the total acoustic noise pollution. Therefore, there is a need to understanding the acoustic noise generation mechanisms in centrifugal compressors, with the purpose of optimizing and finding new noise suppression technologies.

With downsizing the internal combustion engine, turbocharging is desired in order to provide an equivalent power output and an increased specific efficiency of the engine. The turbocharger compound is selected to match the
internal combustion engine operating range. The continuous trend in the automotive industry to decrease the engine size implies for the turbocharging system to operate towards even lower mass flow rate regimes. At such low mass flow rates, flow instabilities are provoked in a centrifugal compressor. Under certain circumstances these instabilities can develop into stalls, rotating stall instabilities, or even the more destructive surge instability. Surge is characterized by severe flow reversal in the compressor, and high amplitude low frequency pressure pulsations. Additional noise generation is associated with turbomachinery operating at such low mass flow rate conditions, as compared with design operating conditions. For example, the interaction of the unsteady flow at this operating condition with the compressor by-pass valve can cause acoustic intensities up to 150 dB, see e.g. Fontanesi et al. (2014).

In general, several different acoustic noise generation mechanisms are provoked in a turbocharger compressor, which can be categorized accordingly to their different acoustic and appearance characteristics Rämml & Åbom (2007); monopoles, dipoles, and quadrupoles. Monopoles are generated by moving volume sources, such as e.g. the shock waves on the compressor blades, which have a tonal radiation characteristic. Especially at high rotational speeds, when the impeller blades move close to the speed of sound, the tonal buzz-saw noise becomes notable. Quadrupole sources are generated by the turbulent fluctuations at high speed flows. The dipole sources are pressure fluctuations on the solid surfaces. An inherent source with uniform inflow conditions is the tonal noise associated with the blade passing frequency (BPF), which is the rotational frequency times the number of impeller blades.

An important additional acoustic noise is the narrowband tip clearance noise, which can become dominant over the tonal blade passing frequency noise, as assessed by Raitor & Neise (2008). It occurs at approximately half of the blade passing frequency and is more prominent at lower compressor speeds. Raitor & Neise (2008) suggested that the noise is generated by secondary flow motion in the gap between the compressor casing and the impeller blades similar to axial compressors, see also Kameier & Neise (1997). Further, a relation between rotating flow instability and the tip clearance noise is hypothesized. Galindo et al. (2015) investigated the influence of the tip clearance size on the acoustic noise generation in a centrifugal compressor, where the variations of the tip clearance have been shown to have insignificant impact on the noise generation at near-surge operating conditions. Other researchers Mendonça et al. (2012) and Tomita et al. (2013) found the occurrence of the narrowband noise at higher frequencies with respect to the rotational speed. Mendonça et al. (2012) relates a rotating instability with this noise source.

Another noise source being more evident at near-surge operating conditions, coined as “whoosh” noise Evans & Ward (2005) or surge noise Teng & Homco (2009), which is described as more broadband noise, stretching over several orders of kHz. Torregrosa et al. (2014) mention the broadband noise to
occur in the range of 800 to 2000 Hz for a similar size compressor as the one used in the present study. Nonetheless, the whoosh noise is more apparent at near-surge operating conditions than at actual deep-surge operating conditions (Evans & Ward, 2005) and Broatch et al. (2015)). Karim et al. (2013) relate an unfavorable incident angle at the leading edge of the compressor blade with the noise generation and Broatch et al. (2015) suggest rotating flow structures in the blade passages as the cause for the whoosh noise. However, Evans & Ward (2005) have found that the main noise source is located further downstream in the compressor outlet hose.

Nonetheless, it remains challenging to associate the perceived acoustic noise with the actual generation mechanism. Therefore, further investigations are required to clearly identify the correlation between flow phenomena and acoustic noise production.

Several experimental and numerical studies have been conducted with the aim of analyzing the origin of the acoustic noise sources in a centrifugal compressor. While acoustic measurements on the actual geometry are most accurate and reliable, the flow instability structures generating the noise are difficult to investigate with experimental assessment tools. This is because of the challenges involved with assessing a confined flow environment. Nevertheless, probe pressure measurements can be carried out which can be used to analyze the compressor flow (Raitor & Neise, 2008) and Torregrosa et al. (2014). Moreover, Particle Image Velocimetry (PIV) measurements can be obtained, currently been restricted to locations upstream of the impeller face (Guillou et al., 2012). Thus, a complete assessment of the flow related mechanisms triggering instabilities inside of the centrifugal compressor is limited when experimental methods are considered.

### 2.4.1. Surge frequency

The surge frequency of centrifugal compressors can with some idealizations be approximated using the Helmholtz resonator theory. In Fig. 2.2 the outlet pipe is considered as a static plenum volume \( V_P \). The compressor volume \( V_C \), i.e. analogue with the Helmholtz bottle neck, a mass \( m \) is being resonated. This volume thus approximates the volume of air in the bottle neck over an equivalent length \( L_C \). In the works by Rothe & Runstadler (1978), the surge frequency is derived and given as follows:

\[
 f_0 = \frac{\omega_0}{2\pi} = \frac{a}{2\pi L_C} \sqrt{\frac{V_C}{V_P}}
\]  

(2.24)

This frequency can be seen as the elapsed time for a perturbation to travel the length \( L_C \) at the speed of sound \( a \), at which the pressure wave propagates. In Eq. 2.24 the plenum volume, i.e. the volume of the outlet pipe, appears in the denominator. This relates with a spring constant of the air in the plenum being inversely proportional to the resonance frequency. If the volume
$V_P$ is reduced, e.g. by considering a shorter outlet pipe installation, then the resonance frequency increases. Several researchers have shown that the Helmholtz resonator theoretical model can give fair predictions of the surge frequency in centrifugal compressors as compared with experimental data, see works by Rothe & Runstadler (1978); Fink et al. (1992). However, due to the complex geometry there is a complication in how to define the compressor equivalent length $L_C$ and also where the plenum starts. The point is that the resonance frequency according to Eq. 2.24 depends on how the equivalent length is defined.
CHAPTER 3

Modeling Compressor Flows

3.1. Challenges

Numerical simulations, opposed to experimental measurements, are well suited to visualize the three-dimensional flow inside centrifugal compressors, and can therefore supplement the analysis. Steady-state Reynolds-averaged Navier-Stokes (RANS) simulations in combination with the so-called Proudman noise source model have been used for preliminary investigations, see e.g. works by Fontanesi et al. (2014). However, the flow at near-surge condition is of pulsating nature and consists of both broadband and narrowband features. Therefore, it is important to utilize a numerical approach that can capture accurately the dynamics and fluctuations associated with these flow regimes. Since it is computationally expensive to resolve all flow scales occurring in the compressor, partial flow modeling is commonly employed. Galindo et al. (2015) compared the pressure fluctuation spectra obtained based on unsteady RANS (URANS) and Detached Eddy Simulation (DES) data and found that more realistic results are obtained with the less restrictive modeling approach DES. With DES, unresolved zones of the flow are modeled by some URANS approach, which are ideally only the near wall regions, while the less dissipative Large Eddy Simulation (LES) is performed in the rest of the domain.

Computation of sheared and free shear flows using the unsteady compressible Navier-Stokes equations is possible. However, the acoustic energy is typically orders of magnitude lower than the energy from the turbulent fluctuating field. This pose challenges with directly resolving the acoustic waves. Another complication is that the acoustic field is many times larger than the turbulent fluctuating field which calls for very large computational domains. Incoming acoustic characteristic in the farfield calls for special treatment of boundary conditions. Boundaries must be appropriate to prevent reflection of acoustic waves, but also in certain regions to provide inflow and outflow of the aerodynamic field.

For an accurate representation of the interaction between the incoming flow and the compressor’s impeller, the computational grid associated with the impeller is rotating at the prescribed RPM, according to the compressor’s operating condition. Moreover, the volute is not symmetric which results in non-uniform flow around the diffuser passage which implies need for 360 degree
geometry. The smallest time and length scales are associated with boundary layer fluctuations and or the blade passing frequency. This calls for a small time-step size as well as highly refined grids. Together with the flow induced flow phenomena triggering surge at dozens of hertz this leads to exceptionally long run times which may be difficult to realize on disposable computer resources. Another complication is that transient LES calculations produce a large amount of data which demand adequate memory storage capacity as well as efficient parallelization of file input and output. The flow in the compressor is also compressible which leads to associated compressibility issues calling for shock capturing techniques and appropriate solver procedures.

3.2. Turbulence modeling and challenges

Instead of resolving all eddy length scales in the energy transfer cascade, time-averaged equations are considered. For incompressible flow the quantities pressure $p$ and velocity $u_i$ are separated into a mean and into a fluctuating part. This is known as Reynolds average:

$$u_i = \bar{u}_i + u'_i \quad \text{and} \quad p = \bar{p} + p'$$

(3.25)

where the over-bar represents the mean part and the prime indicates the fluctuating part. The fundamental concept in turbulence modeling is that the mean values of the fluctuation vanish. For compressible flow, turbulent fluctuations cause density variations. This requires Favre averaging, $\tilde{\phi} = \bar{\rho}\tilde{\phi}/\bar{\rho}$, where the over-bar depicts Reynolds average. In this procedure, the turbulent fluctuations are separated, and the mean flow is density weighed. The averaging of the Navier-Stokes equations leads to new unknown quantities than available equations because of the non-linearity. This is known as the turbulence closure problem.

The fluctuation’s energy is the specific turbulent kinetic energy:

$$K = \frac{1}{2} \left( \overline{u'^2} + \overline{v'^2} + \overline{w'^2} \right)$$

(3.26)

From observations, these fluctuations occur at all scales simultaneously. In accordance with the previous section the smallest turbulent length scales is dictated by the Kolmogorov scale, as compared with the mean free path of molecules which is yet orders of magnitude smaller. The mixing length is the size of the energy carrying eddy structures. In the framework of modeling should be in the order of $10^3\eta$. From the theory there is dissipation of energy where energy is being transferred from larger to smaller scales. For turbulence modeling, description of production, dissipation and transport of turbulent kinetic energy are essential components.
3.2. TURBULENCE MODELING AND CHALLENGES

The time-averaged flow quantities introduced into Eqs. 2.2 to 2.4 results in the compressible Reynolds Averaged Navier-Stokes (RANS) equations:

\[
\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_i)}{\partial x_i} = 0, \quad (3.27)
\]

\[
\frac{\partial (\bar{\rho} \bar{u}_i)}{\partial t} + \frac{\partial (\bar{\rho} \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\sigma}_{ij}}{\partial x_j} + \partial \tau_{ij}, \quad (3.28)
\]

and

\[
\frac{\partial}{\partial t} \left( \bar{\rho} \left( \bar{e} + \frac{\bar{u}_i \bar{u}_i}{2} \right) + \frac{\bar{\rho} u'_i u'_j}{2} \right) + \frac{\partial}{\partial x_j} \left( \bar{\rho} \bar{u}_j \left( \bar{h} + \frac{\bar{u}_i \bar{u}_i}{2} \right) + \bar{u}_j \frac{\bar{\rho} u'_i u'_j}{2} \right) \quad (3.29)
\]

The viscous stress tensor \( \bar{\sigma}_{ij} \) gives transport of momentum through motions at the molecular scale. The Reynolds stress term \( \tau_{ij} = -\bar{\rho} u'_i u'_j \) gives transport of momentum through motion at the macroscale. This term introduces six unknown independent elements, which require modeling for turbulence closure.

3.2.1. Eddy-viscosity modeling of the Reynolds stress

An Eddy-viscosity based modeling approach is commonly adopted based on the works by Boussinesq, where the Reynolds stress terms are related to the mean velocity gradients:

\[
-\rho u'_i u'_j \approx 2 \rho \nu_T S_{ij} - \frac{2}{3} \rho \nu_T S_{kk} \delta_{ij} \quad (3.30)
\]

where the strain rate tensor \( S_{ij} \) is according to Eq. 2.6. It should be noted that \( \nu \) is a property of the fluid whereas the introduced turbulent viscosity \( \nu_T \) is a property of the flow and hence a function of the spatial location. From Eq. 3.30 it is assumed that the Reynolds stress tensor can be determined from single point quantities. It assumes a linear dependence on the mean velocity gradient components in the strain rate tensor and thus independent of rotational influence. These assumptions are not necessarily true for swirling flows in centrifugal compressors but can give reasonable predictions in computational affordable time-frames. There is a range of models for prediction of the transfer of turbulent kinetic energy to heat as well as the turbulent viscosity but some of the more commonly used rely on solving two transport equations for the turbulent kinetic energy and the turbulent dissipation rate. These are known as the \( k-\epsilon \) models. An alternative to the \( k-\epsilon \) approach is the \( k-\omega \) model, where \( \omega \) is the specific dissipation rate and is proportional to \( \epsilon/k \). The derivation of the \( k \) transport equation starts with the momentum equation of \( u_i \) with introduced Reynolds decomposition. Then the Reynolds equation is subtracted, which yields an equation of the velocity fluctuation \( u'_i \), which is subsequently
multiplied by $u'_i$ and averaged. Together with eddy-viscosity modeling (EVM) the transport equation for $K$ and $\omega$ takes the following form:

$$
\left( \frac{\partial}{\partial t} + \bar{u}_i \frac{\partial}{\partial x_j} \right) K = P - C_\mu K \omega + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_T \sigma_T \right) \frac{\partial K}{\partial x_j} \right],
$$

$$
\left( \frac{\partial}{\partial t} + \bar{u}_i \frac{\partial}{\partial x_j} \right) \omega = (C_{\omega 1} - 1) \frac{\omega}{K} P - (C_{\omega 2} - 1) C_\mu \omega^2 + \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_T \sigma_T \right) \frac{\partial \omega}{\partial x_j} \right],
$$

Production $P = -u'_i u'_j \frac{\partial \bar{u}_i}{\partial x_j} = [EVM] = 2 \nu_T S_{ij} S_{ij}$,

Dissipation $\omega = \frac{\nu}{C_\mu K} \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_j}{\partial x_i}$,

Turbulent viscosity $\nu_T = \frac{K}{\omega}$ \hfill (3.31)

The production term $P$ is a mechanism for transport of energy from the mean flow, $\omega$ is a term for dissipation of energy to heat and the last terms are related to transport of redistribution of energy in space. With introduction of EVM the $k - \omega$ model is now loosely coupled with the exact equations and closure coefficients with auxiliary relations are needed for closure. The closure coefficients are typically obtained from experimental observations or from Direct Numerical Simulation (DNS) of idealized flow setups constituting homogenous shear flows and or isotropic turbulent decaying flows, see Wilcox et al. (1998). Some of the advantages with the $k - \omega$ model compared to $k - \epsilon$ is a better response to pressure gradients, which is important for capturing separated flows. It offers a better wall boundary condition, where $K \to 0$ and $\epsilon / K \to \infty$. However $k - \omega$ has problems at non-turbulent interfaces at the boundary layer edge, is unphysically sensitive to free-stream conditions and leads to excessive production $P$ in high shear flows. Menter (1993) amends the boundary layer edge problem by introducing a cross-diffusion term in the $\omega$-equation. Moreover introduced a limit bound for excessive production, which lead to the formulation of the so called Menter SST $k - \omega$ two equation model, which has been used in this thesis.

Curvature correction is a computationally cheap technique to include rotational influence in the $k - \omega$ two equation turbulence model. In the formulation the turbulent energy production term is adjusted according to local rotation and vorticity rates. The correction amends the effects of strong (streamline) curvature and frame-rotation.

3.2.1a. Broadband noise source models. Aeroacoustics is ultimately a transient problem and should in the ideal case be treated with a transient approach. Unfortunately transient calculations such as LES take orders of magnitude longer than steady-state RANS calculations. Resolving acoustic waves requires special considerations for grid spacing and solver settings. In steady-state RANS
3.2. TURBULENCE MODELING AND CHALLENGES

Aeroacoustic calculations, using so-called Broadband Noise Source Models (e.g. Lilley, Curle, and Proudman) are commonly employed which provide an estimate of noise sources through correlations. These models allow tracing flow-generated aeroacoustic sources via surfaces and volumes. They can provide an approximate sound power level (e.g. for design of experiment studies). Moreover, decide where to apply grid refinement prior to a transient run such as LES in order to capture frequency ranges of interest. However, there are limitations of the broadband noise source models. For instance: no indication of frequency of the generated noise, the noise associated with large-scale flow features are unsuitable to be quantified by these correlations. Nevertheless, broadband noise source models generally consider location and strength of monopole, dipole, and quadrupole sources. Monopole acoustic sound sources are associated with displacement of the fluid due to acceleration of the moving surface. The monopole sound waves are typically spread out spherically (e.g. pulsating spheres, sirens). Sound generated by dipole represent force or pressure fluctuations on a surface. They consist of two monopoles of equal strength but with opposite phases and separated by a small distance. Noise from impellers, blowers and fans are examples that falls in this category. The quadrupole sound sources are due to velocity fluctuation in volumes and consist of two similar dipoles but at opposite phase. They have a sense of directionality, where lateral sources are associated with shear stress and longitudinal sources are related with normal stress. Turbulent flows commonly consist of lateral quadrupole sources.

The Proudman noise source model evaluates sound emitted by a homogeneous isotropic turbulent flow where the acoustic power per unit volume is estimate as:

$$ AP = \alpha_c \rho_0 \frac{U^3}{L^{5}} \frac{U^5}{a_0^5}, \quad U = \frac{L}{T} \quad \text{and} \quad \alpha_c = 0.629 $$  \hspace{1cm} (3.32)

where $\rho_0$ – farfield density, $U$ – turbulent velocity, and $a_0$ is the farfield sound speed. From a two equation $k - \omega$ model the turbulent length scale can be estimated with $L = C_\mu \sqrt{k/\epsilon}$ where $C_\mu$ is a model constant, and the turbulent time scale is obtained from the specific dissipation rate $\omega$. The factor $\alpha_c$ is correlated from DNS calculations for isotropic turbulent flow. The total acoustic power per unit volume is given as:

$$ P_{AP}(dB) = 10 \log \frac{AP}{P_{ref}} $$  \hspace{1cm} (3.33)

where $P_{ref} = 1 \cdot 10^{-10}$ W/m$^3$ is the reference acoustic power. Computation of the Proudman acoustic power in centrifugal compressors typically shows amplified intensity levels localized in the vicinity of blade leading edges, which relates with sound generated by dipole sources due to pressure fluctuations on a surface. High levels may also be captured at impeller tip edges. The homogenous and isotropic assumption is not completely accurate for centrifugal compressor
flows, which may therefore lead to unreliable results with Broadband correlation models. However, in the event of available accurate computational or experimental data the correlation factor may be adjusted accordingly.

3.2.2. Large Eddy Simulation

Common observations with prediction of turbulent production from the eddy-viscosity models are no influence of dealignment and rotation, respectively. They are adequate as long as the flow field is approximately isotropic turbulent. These observations cause restriction in the capability of capturing anisotropic turbulence, which become important when the centrifugal compressor approach unstable surge condition. For instance, an accurate description of flow driven instabilities such as boundary layer separation, horse shoe vortices and tip clearance vortices in the rotating impeller calls for resolution of anisotropic turbulent effects. Therefore the three-dimensional, time-dependent large-scale turbulent motion responsible for the fluid mixing in the compressor flow, is preferably computed with the Large Eddy Simulation (LES) approach where scales smaller than the computational grid are modeled. This technique applies a spatial filter to the Navier-Stokes equations by a defined cell width function, defined as $\Delta = V_{cell}^{1/3}$. When the filter is applied to the Navier-Stokes equations these are rearranged into a form that looks similar to the unsteady Reynolds Averaged Navier-Stokes equations (RANS) as follows:

$$\frac{\partial \tilde{\rho} \tilde{u}_i}{\partial t} + \frac{\partial \tilde{\rho} \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\sigma}_{ij}}{\partial x_j} + \frac{\partial \sigma_{SGS,ij}}{\partial x_j}$$ (3.34)

The filtered convection term is modeled by introducing the sub-grid scale (SGS) stresses, defined as:

$$\sigma_{SGS,ij} = -\bar{\rho}(\tilde{u}_i \tilde{u}_j - \bar{u}_i \bar{u}_j)$$ (3.35)

The filtered viscous stresses are written as:

$$\bar{\sigma}_{ij} = -2\mu \tilde{S}_{ij} + \delta_{ij} \frac{2}{3} \mu \tilde{S}_{kk}$$ (3.36)

A linear relation is assumed between the SGS stresses and the resolved strain rate and is obtained with an eddy-viscosity closure known as the Boussinesq approximation:

$$\sigma_{SGS,ij} = -\mu_t \tilde{S}_{ij} + \delta_{ij} \frac{2}{3} k_{SGS}$$ (3.37)

where $\mu_t$ and $k_{SGS}$ are the SGS turbulent viscosity and kinetic energy, respectively. LES needs closure by modeling of the SGS stress tensor either explicit or implicit.

In the numerical procedure, the governing equations are considered in continuous integral form and then formulated in discrete form in finite control volumes that constitutes the computational domain of the compressor. Via Taylor series expansion an approximate algebraic representation is obtained for the transient term, which is discretized with a second-order temporal scheme.
3.3. ANALYSIS OF COHERENT FLOW STRUCTURES

The convective flux, the diffusive flux and the source terms, respectively are discretized with a bounded third-order spatial scheme. The discretization introduces unwanted truncation errors since higher order terms in the infinite Taylor series expansion are omitted. However, a finite volume discretization (i.e. second-order or higher) exhibits low dissipative error and it has been demonstrated adequate in conserving the kinetic energy in the turbulent cascade. Further, the dissipative truncation error stresses mimic Smagorinsky-type SGS modeling for the energy equation, see works by Margolin & Rider (2002) and Fureby & Grinstein (2002). The role of the SGS model is to dissipate enough kinetic energy at the smallest sub-grid scale. Since this effect is obtained from the truncation error, implicit LES with no explicit SGS is considered and hence no explicit filter is required. This approach limits uncertainties from introduced SGS modeling constants and is therefore preferred.

Through dimensional analysis the Kolmogorov spectral energy density scalar for a velocity fluctuating component, with dimension kinetic energy per wavenumber, is given as:

$$E = \alpha \epsilon^{2/3} \kappa^{-5/3}$$  

(3.38)

where \(\alpha\) – constant, \(\epsilon\) – dissipation rate, and \(\kappa\) – the wavenumber. The scalar spectral density based on a pressure fluctuating quantity is obtained in a similar manner:

$$E_{pp} = \alpha \epsilon^{4/3} \kappa^{-7/3}$$  

(3.39)

The validity of the implicit LES approach can subsequently be evaluated by verifying the power spectral density of a fluctuation quantity in a point exhibiting isotropic turbulence and analyzing the characteristic energy decay slope in the inertial subrange (i.e. \(-5/3\) for velocity and \(-7/3\) for pressure).

3.3. Analysis of coherent flow structures

The turbulent flow is rotational and consist of large coherent motion of vorticity which is defined as the curl of the velocity vector field \(\nabla \times u_i\). Computing the curl of the momentum equation results in the vorticity equation given as:

$$\frac{\partial \omega}{\partial t} + (u_i \cdot \nabla) \omega = (\omega \cdot \nabla) u_j - \omega \nabla \cdot u_j + \frac{1}{\rho} (\nabla \rho \times \nabla p) + \nu \nabla^2 \omega$$  

(3.40)

The last term on the right hand side represents viscous dissipation at small scales. It has a small influence on large coherent structures in high Reynolds number flows. The third baroclinic term on the right generates torque due to acceleration in a flow subjected to density and pressure gradients. It is a mechanism for vorticity generation. The second term on the right describes stretching of vorticity due to compressibility, which follows from the continuity equation. The first term on the right incorporates stretching or tilting of vorticity due velocity gradients.
For centrifugal compressors fitted with a bell mouth inlet streamlines converge towards the impeller. The inlet flow accelerates, the axial velocity increases, and the pressure decreases upstream of the impeller. This implies an expanding air flow with $\nabla \cdot u_j > 0$. Vorticity follow Kelvin’s theorem, i.e. $|\vec{\omega}|$ increase as vorticity tube cross section decreases. In the diffuser, the pressure increases, the velocity decreases, and the flow is compressed, i.e. $\nabla \cdot u_j < 0$.

The vortex stretching term $(\omega \cdot \nabla)u_j$ governs the energy transfer in the turbulent cascade. If two points in the vortex are considered then in statistical average the distance between them in the longitudinal direction will grow over time. This leads to increased vorticity magnitude in the stretched direction and reduced vorticity magnitude in the two orthogonal directions. As the vortex is stretched, its angular momentum is conserved and is proportional to $\omega r^2$, where $r$ is the vortex radius. The kinetic energy associated with the vortical motion scales as $\omega^2 r^2$. For a conserved angular momentum this leads to a reduced kinetic energy where energy is being transferred from large to small vortical structures. Use of the so called $\lambda_2$ criteria is one possibility to extract some of the coherent vortical structures in the flow field, see Fig. 3.1. In the figure some vortical structures are captured: at the tip clearance, at the leading edge, horse shoe vortices at the blade/hub and structures at the diffuser due to boundary layer separation. The structures on the blade surface are related with the strong velocity gradient in the boundary layer. The $\lambda_2$ criteria is a scalar quantity defined by the second eigenvalue of $(S^2_{ij} + \Omega^2_{ij})$ when sorted from minimum to maximum, where $S_{ij}$ is the strain-rate tensor and $\Omega_{ij}$ is the spin tensor. Assessment of the instantaneous flow field using the $\lambda_2$ criteria, as exemplified in Fig. 3.1, is rather intricate to interpret due to the inherent
3.3. ANALYSIS OF COHERENT FLOW STRUCTURES

3.3.1. Flow mode decomposition

Proper Orthogonal Decomposition (POD) was introduced into fluid dynamics applications to extract large energetic flow structures from small-scale turbulent background flows. It is an effective approach to filter out high frequency incoherent turbulent fluctuations from the flow field. POD analysis can be performed in industrial applications to extract characteristic flow features. A short introduction is given by Lumley (2007). In the POD method, eigenfunctions of the spatial (or temporal) correlation matrix, constructed from a series of snapshots of scalar or vector fields, are computed. Thereby, the coherent motion of the flow is extracted, where the modes are favored accordingly to their energy content. One drawback with the POD method is the temporal representation of the modes, since a mode is not associated exclusively to a particular frequency. POD decompose the flow field into temporal $a_j(t)$ and spatial $\phi_j(x)$ components:

$$ u(x,t) \approx a_0(t)\phi_0(x) + \sum_{j=1}^{\infty} a_j(t)\phi_j(x), \quad (3.41) $$

where the zeroth mode represents the averaged mean flow.

Dynamic Mode Decomposition (DMD) is another technique to decompose the flow into modes with an associated frequency, see (Schmid (2010)). DMD is preferred when tonal behavior of the flow is expected, as described in the works by Alenius (2014). The method also provides the growth rate $\beta_j$ at a specific frequency $\omega_j$ and therefore offers a stability criteria for a spectral mode shape. In the DMD procedure, according to Schmid (2010), a non-linear system can be approximated with a linear operator, i.e. the so-called Koopmann operator. The linear operator operates on a sequence of state vectors, which is constructed from the observed time-dependent flow field, e.g. snapshots of velocity or pressure. It is assumed that a linear operator exists and contains the temporal information of the dynamic flow process. The dynamic modes $\phi_j$ are found by computing the eigenvectors of the linear operator. The Frequency is given by:

$$ \omega_i = \frac{1}{\Delta t} \tan^{-1} \left( \frac{\text{Im}(\lambda_j)}{\text{Re}(\lambda_j)} \right) \quad (3.42) $$

where $\lambda_j$ is the eigenvalues of the linear operator. The growth rate is given by:

$$ \beta_j = \ln(\text{mag}(\lambda_j))/\Delta t \quad (3.43) $$

Reconstruction of the mode in time is obtained using:

$$ u_j(x, t) = \text{Re} \left( \eta_j e^{(\beta_j - i\omega_j)t} \phi_j(x) \right) \quad (3.44) $$
where $\eta_j$ denotes the amplitude of the $j^{th}$ mode. The sampling frequency for computation of the DMD modes, i.e. $1/\Delta t$, should be chosen so that the linear operator adequately approximates the dynamic flow process. According to the sampling theorem, aliasing is avoided if there are at least two samples per period. However, Schmid (2010) recommends a sampling frequency of approximately a factor six times the highest frequency of interest.

### 3.3.2. Acoustic pressure perturbation

There are several ways to visualize the acoustical pressure and pressure perturbation. Two commonly used methods are the $p'$ and the Wavenumber decomposition method, respectively. The first method involves computing the scalar quantity $p' = p - \bar{p}$, where the mean pressure $\bar{p}$ is subtracted from the instantaneous pressure. The left image in Fig. 3.2 shows dominance of turbulent fluctuation inside the compressor. Thus the near field is subjected to a high energy level. But as the energy level falls towards the upstream far-field, acoustic waves become distinctively visible with relatively clear wave length, and are seen to propagate at the speed of sound. Since the acoustic farfield consist of different acoustic waves at different wave lengths it is interesting to compute their overall group velocity. This is obtained using the Wavenumber decomposition technique. A line spatial Fourier transform is evaluated to separate acoustical and convective energy content:

$$F(k, \omega) = f^*(k, \omega) f(k, \omega), \quad \forall k \in K, \omega \in [-\infty, \infty]$$ (3.45)

where $f^*$ is the complex conjugate of a real value function $f$, $k$ is the wavenumber and $\omega$ is the frequency. The right image in Fig. 3.2 shows the spectra evaluated along a line from the impeller source to the upstream hemisphere, i.e. in the farfield. The slope of the distinctive line in the wavenumber-frequency plot is used to calculate the group velocity of the disturbances traveling along the line. For this particular compressor, the group velocity is estimated as:

$$\frac{\partial \omega}{\partial k} = \frac{2\pi \cdot 11e3/200}{345} = 345 \text{ m/s},$$

which relates to the speed of sound at approximately ambient upstream conditions.

### 3.3.3. Fourier transform

The Fourier spectra involves a time history signal which is Fourier transformed into a representation in the frequency domain. It allows identifying modes in the signal. The Fourier transform is defined as:

$$FT_t[h(x, t)] = \int_{-\infty}^{\infty} h(x, t)e^{-i\omega t}dt = g(x, \omega), \quad \forall \omega \in [-\infty, \infty]$$ (3.46)

where $h(x, t)$ is a real function which depends on the position $x$ and time $t$, and $\omega$ is the angular frequency. In practice the integration is over a given time period and altered to be periodic which leads to a finite Fourier transform for a specific time block. One benefit with the Fourier transform is that acoustic
3.3. ANALYSIS OF COHERENT FLOW STRUCTURES

Figure 3.2: Side and front view sections in a centrifugal compressor colored with $p'$ with a narrow range to detect sound waves propagating in the upstream farfield of the bell mouth (left). Wavenumber-frequency spectra for computation of the group velocity (right).

and flow field behavior can be characterized in two ways. First, is to determine narrowband and broadband features at discrete points in the computational domain. The spectra in some of these points may indicate tonal behavior and hence give the first sign of an interesting feature. Secondly, Fourier surface spectra is performed to give spatial distribution and filtered to include the desired discrete frequency values or frequency bands. This allows analysis of for instance the power spectral density distribution at frequency bands identified by the point spectra. The ability to contour power spectra distributions enables an understanding of where narrowband flow features originate and how acoustical sources propagates in the farfield.
4.1. Discretization

The governing equations (2.2, 2.3 and 2.4) are a system of non-linear partial differential equations and to the author’s knowledge have no known general analytical solution for unsteady flows in complex geometries such as in a centrifugal compressor. However, an approximate solution can be obtained by solving these equations with an appropriate numerically procedure. The procured starts with deriving transport equations of the mass, momentum and energy conservation equations which are subsequently solved based on for instance the finite volume approach, see work by Ferziger & Peric (2012). The transport equation in continuous integral form represents transport of a scalar quantity in a continuum and includes a transient term, a convective term, a diffusive flux term and a volume source term. The subsequent step involves applying the transport equation to a cell-centered control volume. From there on each term is approximated with a discretization scheme of choice yielding the transport equation in discrete from. In steady-state RANS calculations the transient term is omitted but in transient LES calculations throughout in this thesis an implicit second-order temporal discretization scheme is employed. For the convective term a hybrid MUSCL 3rd-Order/CD as implemented in the CFD code STAR-CCM+ (2015) developed by CD-adapco is used. In its formulation the scheme employs a blend between 3rd-order upwind and 3rd-order central-differencing. The discretization employed to the transport equation results in an algebraic system of the linear equation system being obtained for the transported variable $\phi$ at iteration $k + 1$:

$$a_p \phi_p^{k+1} + \sum_n a_n \phi_n^{k+1} = b$$

(4.47)

The summation is done over all the neighbors $n$ of cell $p$, $b$ – explicit right hand side obtained at iteration $k$, and coefficient $a_p$ and $a_n$ results from discretized terms. Because this system can be very large it is impractical to invert the coefficient matrix on the left hand side by means of Gauss elimination or LU decomposition to obtain the transported variable at the next iteration. Fortunately, the coefficient matrix is typically sparse, therefore, this system is
4.2. NUMERICAL GRID

solved implicitly, in an iterative fashion, preferably using the algebraic multigrid method. By identifying Eq. 4.47 as $A\mathbf{x} = \mathbf{b}$, iterative methods involve a given approximate solution $\mathbf{x}^k$ and then iteratively obtaining a better approximation $\mathbf{x}^{k+1}$. Jacobi and Gauss-Seidel are two common iterative methods which involve visiting each cell in sequence, update the value of $x_i$ in each cell using the information for neighboring cells. The convergence rate of Gauss-Seidel is faster as compared to Jacobi and is therefore preferred. It can also be shown that the total work per iteration with Gauss-Seidel scales as $N^2 \ln N$ which is impractical for large grids. The algebraic multigrid method is a general procedure to further accelerate convergence. Gauss-Seidel provides good smoothing of local errors but convergence is slow since it takes time for boundary information to propagate into the domain. However, on a coarse grid the boundary information propagates faster as compared to a fine grid. This leads to the concept of alternating the Gauss-Seidel iteration on coarse and fine grids and using a transfer operator for interpolating the solution between grids. On fine grids the error is experienced having low frequency but on coarse grids the error is experienced having high frequency. The end result with algebraic multigrid is improved total work per iteration, in the order of $N \ln N$.

A necessary condition for stability is that the domain of dependence of the finite volume discretization should include the domain of the dependence of the partial differential equations. This implies that the information should not travel more than one cell length per time-step and is dictated by the Convective Courant number.

$$CFL = \frac{U \Delta t}{\Delta x} \leq 1 \quad (4.48)$$

where $U$ is a characteristic velocity, $\Delta t$ is the time-step size and $\Delta x$ is a characteristic cell size. Generally, there is some optimal value of Courant number that balances the total number of iterations against the time per iteration. For rotating machinery problems, the blade tip often represents a worst-case condition, which gives a conservative time-step estimate as:

$$\Delta t = \frac{(\Delta x)_{tip}}{\omega R_{tip}} \quad (4.49)$$

where $\omega$ is the rotor speed.

4.2. Numerical Grid

The computation domain is discretized in finite control volumes. Due to the geometrical complexity (e.g. high surface curvature at leading edges of the impeller blades as well as thin gaps and tight corners) unstructured polyhedral grids are considered. Some of the advantages are that the polyhedral mesh generation process is more easily automated in comparison with structured curvilinear grids and it may exhibit less numerical dissipation of some wake flows than Cartesian hexahedral body-fitted meshes.
The near wall region over non-slip walls is resolved with prismatic cell layers due to the large wall normal velocity gradient. In upstream and downstream piping, wall functions are adopted since the wall bounded flow is approximately attached and steady. However, towards off-design condition the boundary layer may separate from the impeller surface and therefore such boundaries are resolved with no use of wall functions. Thereby, a more accurate prediction of potential separation points may be obtained. To accommodate well-conditioned boundary conditions the grid is gradually stretched towards the openings of the computational domain. To handle the impeller rotation and capture the blade passing relative to other components, the domain is divided in a stationary region and a rotating region with a sliding interface between. The two sides remain implicitly coupled and the cell connectivity is recalculated every time-step. Fluid information in time and space are interpolated to preserve flux continuity across the interface to avoid introducing spurious perturbations to the flow field. The mesh of the non-stationary part is made to rotate at an angular velocity $\omega$ about the compressor axis. This allows for Coriolis and centrifugal force terms in the transport equations in the algebraic representation of the governing equations.

Particular grid refinement considerations are necessary to ensure that the frequency range in both flow and acoustic fields are adequately captured. For this, the grid frequency cutoff index gives some guidance. This is motivated from the concept that the turbulent kinetic energy $k$ relates to the velocity fluctuations. The minimal cell count to represent an acoustic eddy is approximately two cells per direction. Therefore, the maximum frequency $f_{MC}$ reasonably resolved by the local grid spacing $\Delta x$ is:

$$f_{MC} \approx \frac{\sqrt{\frac{2}{3} k}}{2\sqrt[3]{\text{cell}}}$$  \hspace{1cm} (4.50)

In steady-state RANS calculations the grid in the rotating impeller region is not moving. Instead, a relative mesh rotation is simulated by activating rotational momentum source terms in the governing equations at cells within the specific rotating region. The relative motion accounts for transformation and exchange of velocities on either side of the dividing interface. Provisions are made for circumferential averaging of conserved quantities. An implicit connection is employed on the cells on either side of the interface. Since these calculations are steady-state, the blade passing effects are not included (e.g. interaction of trailing edge wake shedding with the volute). Results can be as good (or bad) as the momentum source terms applied.

4.3. Boundary Conditions

For centrifugal compressors stationary in space, the upstream farfield fluid flow may be assumed to be approximately quiescent with known quantities of
4.3. BOUNDARY CONDITIONS

the stagnation temperature and pressure, respectively. However, the impeller rotation introduces hydrodynamic turbulent fluctuations at significant energy levels, which manifest as both broadband and narrowband features in the spectra. Pressure waves traveling in the upstream direction at the speed of sound from the impeller are generated. For a correct treatment the inlet boundary is therefore considered non-reflective for incoming sound waves by continuous adjustment of the target cell face Mach number, as well as static temperature and pressure, respectively. This is obtained by extrapolating from neighboring cell centered values and correcting the boundary face fluxes for transported one-dimensional disturbances. The freestream velocity on the boundary for subsonic inflow is adjusted by a combination of the boundary normal velocity and the freestream velocity as follows:

\[
\vec{v}_f = \vec{v}_\infty + (v_{fn} - v_\infty) \frac{\vec{a}}{|\vec{a}|}
\]  

(4.51)

where \(v_{\infty n}\) is the boundary normal freestream component. The boundary normal velocity \(v_{fn}\) is obtained from characteristics:

\[
v_{fn} = \frac{1}{2}(v_{0n}^r + v_\infty) + \frac{c_0 - c_\infty}{\gamma - 1} \tag{4.52}
\]

where \(v_{0n}^r\) is the boundary normal velocity component extrapolated from the neighboring cell center. The speed of sound \(c_\infty\) is obtained from the freestream temperature. \(c_0\) is the speed of sound obtained from the boundary temperature extrapolated from the cell:

\[
c_0 = \sqrt{\gamma R T_0} \tag{4.53}
\]

For subsonic inflow, the boundary pressure is obtained from the freestream pressure via the isentropic relation:

\[
p_f = p_0^r \left(\frac{T_f}{T_0^r}\right)^{\gamma p / R} \tag{4.54}
\]

The boundary face temperature is obtained as:

\[
T_f = \frac{c_f^2}{\gamma R} \tag{4.55}
\]

where:

\[
c_f = \frac{1}{2}(c_\infty + c_0) + \frac{1}{4}(\gamma - 1)(v_{0n}^r - v_\infty) \tag{4.56}
\]

Where the boundary is not normal to the incident compression waves, errors can occur in the non-normal directions. For an implementation of non-reflecting boundaries see works of Giles (1990) and Saxer (1992).

In experimental setups the outlet mass flow rate is commonly regulated with a throttling valve. Since a fixed mass flow is applied at the outlet, temperature and pressure are extrapolated from upstream characteristics when the flow is purely outgoing. At low mass flow, as approaching off-design condition
(e.g. surge) the compressor is subjected to flow reversal. To ensure robustness under such conditions the outlet is positioned relatively far downstream with a gradual grid stretching. This helps suppress potential backflow and stabilize the numerical approach, yet flow reversal in the vicinity of the volute exit cone is allowed. A non-reflecting treatment could be considered also at the outlet but since the surge condition is unstable there would be no explicit mechanism to prevent the operation from drifting away from this condition. The adopted grid stretching causes an elongation of vortical structures where the tangential component is damped. This introduced numerical viscosity assists the damping of pressure reflection. Adiabatic non-slip walls are considered in the compressor.
CHAPTER 5

Cases and Results

5.1. Setup & Case descriptions

Steady-state, three-dimensional Reynolds Averaged Navier-Stokes (RANS) simulations are used to explain differences in the compressor performance maps for two automotive centrifugal compressors of different sizes. Special emphasis is put on conditions closer to the surge line, at different speed lines. The two investigated compressor geometries and the corresponding specifications are shown in Fig. 5.1. One is named small compressor (diffuser area ratio = 0.56) and the other is named large compressor (diffuser area ratio = 0.62). The surge by-pass channel with a lock valve and the ball bearing features have been removed and the impeller nut is idealized for simplification of the simulation setup. The inlet and volute exit pipes are extruded several diameters in the boundary normal directions, where the piping systems are equal for both compressors. Hence, an idealized installation of the compressor is considered (replicating the gas stand experiment) under so-called cold conditions with adiabatic wall boundary conditions.

In order to characterize the flow behavior occurring in the small and large compressors, a large number of operating conditions need to be simulated (different rotational speeds and mass-flow rates). Computational techniques based on the steady-state RANS formulation are relatively fast, robust, and affordable as compared with unsteady approaches Baris & Mendonca (2011), which can capture the mean flow features and trends corresponding to the measured global performance parameters. Despite their limitations, steady-state RANS modeling can be utilized for covering an impressive number of cases within the stable compressor operating conditions, in a relatively short computational time Sundström et al. (2014, 2015b).

Industry-standard modeling is employed to provide fast (3 hour) turn-around for each compressor operating point on a 6 core CPU workstation. The flow governing equations are solved by a compressible implicit coupled solver STAR-CCM+ (2015), which is based on the finite volume approach. The two-equation $k – \omega$ SST turbulence model Menter (1994) is employed. About one million polyhedral computational cells are used to discretize the computational volume of the two compressors. Two prism-layers are considered towards the no-slip walls. The Moving Reference Frame (MRF) approach is considered to
Figure 5.1: The small and the large compressors, respectively, are shown from top to bottom. Side and front views for each compressor are presented. The mesh resolution is indicated in the enlarged insets.

take the effects of impeller’s rotation into account. Moreover, mixing-plane interfaces are used which gives a circumferential averaged flow at the interfaces between the impeller region and the stationary. This treatment allows for mean radial variations.

Numerical analysis has also been performed with a ported shroud centrifugal compressor. Figure 5.2 shows side and front views respectively of the CAD geometry together with the location of monitoring points and planes, used for data sampling and statistics purposes. Four ribs support the ported shroud in an asymmetric arrangement. The ported shroud technology allows some flow to recirculate back from the impeller to the compressor inlet. Thereby, the operating range of the compressor is widened near the surge-line Guillou et al. (2012). Figure 5.1 and Fig. 5.2 tabulates some specifications in the text annotations for each compressor.

The geometrical features associated with the gap between the impeller and the diffuser back-plate, as well as the oil bearings are not available in the provided CAD file for the ported shroud compressor. Thus, the small cavity between the impeller and the back-plate has been omitted in the present simulations. Polyhedral cell types are used to discretize the computational domain in finite control volumes for the three different compressors. Local refinements are used to resolve the flow in important areas, i.e. around the
5.1. SETUP & CASE DESCRIPTIONS

Figure 5.2: Locations of plane sections and monitoring points are indicated for the ported shroud compressor. Side and front views are presented. The mesh resolution is indicated in the enlarged inset.

impeller’s blade edges and at the volute’s tongue. Additional refinements are applied in the open ports, and shroud cavity for the ported shroud compressor. Further, refinements towards non-slip walls are performed using aligned prism cells. The benefit of polyhedral and prism cells is that the mesh can be generated automatically in a complex geometry, such as the compressor. Because the polyhedral cell has more cell faces than for instance hexahedral or tetrahedral cells, it is less numerical diffusive with a Finite Volume (FV) face based solver in rotational flows, see works by Chow et al. (1996). However, far upstream and downstream towards the inlet and outlet boundaries respectively, the flow is more aligned. In those areas it is numerically beneficial to use aligned cells such as the hexahedral cells. Therefore the mesh at the bell-mouth inlet is extruded in the radial direction by 3*De, with De being the exducer diameter. The exit pipe from the volute is extruded by 10*De in the flow direction normal to the outlet. Note that the inlet and outlet extrusions as well as boundary conditions used for the ported shroud compressor are only approximations to the measurement rig and the experimental setup at the University of Cincinnati. Thus, a perfect match with experimental measurements (Guillou et al. (2010); Hellstrom et al. (2010)) cannot be expected.

Another benefit with such extrusions is that they allow a gradual increase in cell aspect ratio towards the openings (inlet or outlet). Thereby, eventual reflections of pressure waves from the inlet or the outlet boundaries of the
5. CASES AND RESULTS

computational domain can be damped. Critical areas, such as the blade leading and trailing edges, and the volute tongue, have highly curved surfaces and the underlying geometry is better preserved with the polyhedral cells. However, because the polyhedral cell has more faces than other cell types it requires more computational resources. Nevertheless, since this cell type is less diffusive, an equivalent accuracy can be obtained with fewer cells.

The operating conditions considered for the ported shroud compressor are listed in Table 1. A total inlet temperature of 296 K is used for all cases, i.e. Design, Near Surge and Surge operating conditions for two different speedlines. Because a large hemispherical inlet boundary is used the total inlet temperature is almost identical with the static inlet temperature. Note, for the LES method at near surge and surge conditions a target static temperature is specified instead of the total temperature. For the steady-state RANS simulations the turbulence intensity is set to 1% and turbulence viscosity ratio is set to 10 for all cases. The simulations are run until a statistically converged solution is obtained.

<table>
<thead>
<tr>
<th>Cases</th>
<th>Operating condition</th>
<th>( \dot{m} ) (kg/s)</th>
<th>( \omega ) (rpm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Surge</td>
<td>0.05</td>
<td>64000</td>
</tr>
<tr>
<td>B</td>
<td>Near-Surge</td>
<td>0.085</td>
<td>64000</td>
</tr>
<tr>
<td>C</td>
<td>Design</td>
<td>0.28</td>
<td>64000</td>
</tr>
<tr>
<td>D</td>
<td>Near-Surge</td>
<td>0.166</td>
<td>88000</td>
</tr>
<tr>
<td>E</td>
<td>Design</td>
<td>0.28</td>
<td>88000</td>
</tr>
</tbody>
</table>

Table 1: Operating Conditions considered with the ported shroud compressor.

5.2. Assessment of Accuracy

Accuracy assessment is important for all kinds of numerical simulations since the computed result depends on several modeling assumptions and the resolution of the employed discretization. There are mainly two categories for this, one refers to verification and the other refers to validation, see works by Roache (1998). Verification involves adequate measures to ensure that implemented methods and routines in the CFD code are computing expected results. Simplifications of the governing equations, by considering e.g. steady-state, 2D and incompressible flow and no heat transfer in idealized geometries such as the flow in axisymmetric straight pipes have exact analytical solutions. The CFD code is subsequently setup for such flows and the computed result is verified against the exact analytical solution. For unsteady flows in complex geometries (e.g. centrifugal compressors) there is no known general solution of the governing equations. This leads to the second category, namely validation. Since the result from the employed numerical approach depends on the grid
resolution a number of grids with successively finer resolutions are tested and compared. A fine grid yields less numerical diffusion and resolves more features in the flow whereas a coarse grid is affected by more numerical diffusion and therefore does not resolve as many features in the flow. In numerical simulations there are truncation errors associated with the discretization scheme, the resolution of the grid, and round off errors associated with the finite floating point arithmetic precision in the computer. Consistent discretization schemes should be considered, which results in a truncation error that goes to zeros as the grid spacing tends to zero $\Delta x \to 0$. The finite precision sets a lower limit on the grid spacing since round off errors will start to accumulate. In practice computational resources, including data storage is limited. This sets a limit on the grid resolution. Therefore truncation errors are typically more discernible compared to round off errors. A common strategy in grid dependency assessment involves halving the cell edge length in a uniform fashion between three different grids. Since the cell count grows exponentially its quickly leads to unaffordable computational costs. A computational cheaper strategy is to assess if the grid captures some of the more relevant features in the flow as compared with available experimental data. Thereafter, only refine areas subjected to large gradients.

5.2.1. RANS solver, validation

The global performance prediction in terms of the total-to-total pressure ratio (PR) and total-to-total efficiency are shown in Fig. 5.3 for the small and the large compressor. These are commonly calculated as follows:

\[
\text{Pressure ratio, } PR = \frac{p_{02}}{p_{01}} \quad (5.57)
\]
\[
\text{Efficiency, } \eta = \frac{(\gamma-1)}{(\gamma-1)} \left( \frac{T_{02}}{T_{01}} - 1 \right) \quad (5.58)
\]

where $\gamma$ represents the specific heat ratio of 1.4. Index 0 denotes stagnation pressure or temperature. The indexes 1 and 2 refer to quantities at the inlet and the outlet, respectively. Equation 5.58 is an estimate of the efficiency assuming isentropic flow.

A perfect match of the predicted values with the experimental measurements is not expected, since the simulated geometry is not identical with the measured geometry. Nevertheless, the PR shows overall a good trend apart from a more pronounced humpback feature at peak PR at the higher speedlines. There can be diverse grounds for this. First there is a geometrical difference where the surge by-pass channel is not included in the simulation, so there is a possibility that this geometrical feature yields some additional losses at the higher speedlines. The steady-state RANS convergence characteristics deteriorate as compressor operation approaches stall at low mass flow rates, which is due the onset of large-scale flow instabilities.
The total-to-total efficiency also shows an overall good trend with the experimental data although the levels are over-predicted in the simulation for both the small and the larger compressor. The agreement of the absolute levels is seen to improve with increased mass flow rates. Since the larger compressor operates at higher mass flow rates than the small compressor the agreement is seen to be better with the larger compressor. The reason for the over-prediction in absolute levels is that the boundary conditions are not identical between the simulation setup and the experimental setup. In the simulation setup, adiabatic walls are assumed. This is not the case in the experimental setup where the turbocharger is running under so call hot conditions. Therefore the omission of wall heat transfer may have some significance in the over-prediction at low mass flow rates, but at higher mass flow rates the assumption of an adiabatic wall improves, see work by Serrano et al. (2013).

Figure 5.3: Compressor pressure ratio and efficiency vs. corrected mass flow (small compressor at the top row and the large compressor at the bottom row). The speed line is indicated as the $u_{redC}$. Normalization of corrected mass flow rate and speed lines, for the small and large compressor, is done for propriety reasons.

5.2.2. LES solver, validation

LES computed pressure ratio as well as the efficiency for the ported shroud compressor in Fig. 5.2 are compared quantitatively with experimental data in
5.2. ASSESSMENT OF ACCURACY

Fig. 5.4. Equation 5.58 was used to compute the efficiency from the experimental data (Guillou et al. (2012, 2010); Hellstrom et al. (2010)) and is used here for a corresponding comparison. LES data is shown for the fine grid resolution. The Experimental data are displayed with dots and LES data with a cross. The general trend indicates that the numerical setup is reasonably close with the experimental setup and can therefore produce fair predictions with respect to global performance parameters. The vertical bar indicates time variation in monitored PR and $\eta$ estimations, respectively. The horizontal bar represents the variation in mass flow rate. In near optimum design condition the time history signal for Case C shows small variations since the flow field is approximately steady. Case B shows small periodic variations, but may be considered as approximately stable, because sudden perturbations tend to decrease in amplitude with time. As conditions approach surge (i.e. Case A) the variation increases and the mass flow is observed to oscillate back and forth, but with no net backflow. A wave-like character with eight periods is observed in the PR time history signal at approximately 43 Hz. This corresponds to the surge frequency and it is in the same order of magnitude with the Helmholtz resonator frequency in Eq. 2.24. The sample length considered for Case A and Case B are relatively short, so the signal lacks resolution in the low frequency range but can be improved with a more generous run time. Despite a good overall trend there are some differences compared with the experimental data. Some of the main sources for the difference are the challenges with exactly matching the boundary conditions as well as using an identical replication of the geometry used in the experimental setup. There are also uncertainties associated with the numerical accuracy, which includes discretization errors of the integral formulation of the flow governing equations in finite volumes using an implicit 2nd order temporal scheme and an implicit 3rd order bounded central difference scheme in space. For example, a 2nd order scheme may have a leading odd-order derivative term being truncated and would hence be associated with dispersion errors. Such an error may introduce phase errors and can contribute to some of the difference in exactly matching the surge frequency obtained experimentally. On the other hand, a 3rd order scheme may have a leading even-order derivative term. Those terms may be associated with a dissipation error and could contribute to amplitude errors.

An important task for dynamic simulations is to evaluate the relevance of the numerical approach and assess that there is a sufficient numerical grid resolution employed which captures the most relevant features in the flow. Three grids named Coarse, Medium and Fine are used for this purpose and run for the reference operating condition $\omega = 64000$ rpm at near maximum efficiency condition $\dot{m} = 0.28$ kg/s. The average cell edge length in the Coarse grid is successively reduced a factor of two, which yields the two finer grids. The characteristics of the grids used for the ported shroud compressor are listed in Tab. 2. Implicit time-stepping does not require that the convective
Figure 5.4: a) Global performance parameters, pressure ratio (PR) and Efficiency $\eta$ from LES and Experiment for two different speed lines. b) PR time history signals for Case A, B and C. The black curve corresponds to Experimental data and the gray curve to LES data. The pressure ratio is computed from the mass flow average pressure between a plane located two diffuser diameters downstream of the volute exit, and a curved plane located one diffuser diameter upstream from the bell mouth.

Courant number is unity everywhere in the domain, but for accurate time dependent solutions the CFL condition should be fulfilled. This condition is of particular importance to ensure that information propagates correctly across the sliding interfaces between the rotating impeller region and with surrounding stationary regions. For a fair comparison the time-step size is adjusted so that the convective Courant number is similar between grids.

<table>
<thead>
<tr>
<th>grid</th>
<th>cell count</th>
<th>$\Delta x_i$ [mm]</th>
<th>$\Delta t$ [deg/time-step]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>0.7 million</td>
<td>2.8</td>
<td>4</td>
</tr>
<tr>
<td>Medium</td>
<td>2.8 million</td>
<td>1.4</td>
<td>2</td>
</tr>
<tr>
<td>Fine</td>
<td>9 million</td>
<td>0.7</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 2: Characteristics of grids. The cell count and the average cell edge length $\Delta x_i$ are listed. The last column shows the chosen time-step $\Delta t$ used for a certain grid and is given as the number of degrees of impeller rotation per time-step.
5.2. ASSESSMENT OF ACCURACY

The grid sensitivity procedure employed is based on Richardson extrapolation and suggest to state the quality in terms of several different parameters, see Celik et al. (2008). Figure 5.5 presents the comparison between the static pressure solution at two different radii on the compressor back-plate in the diffuser and the corresponding experimental data. Solutions from the three different grid resolutions are depicted. The data is time averaged for a total of 20 impeller revolutions. The solution from the Medium and Fine grids lays almost on top of each other whereas the Coarse grid shows a poorer trend at peak pressure at 3 o’clock corresponding to a volute angle of approximately 25 degree. Possible reasons for having differences between the experiment and the simulations are due to non-identical boundary conditions as well as simplifications of the geometry.

![Figure 5.5: Static pressure distribution in the diffuser on the compressor's back-plate at two different radii $r = 55.5$ mm and $72.5$ mm for Coarse, Medium and Fine grids. The inset indicates locations of measurement probe points on the back-plate.](image)

The differences between grids causes a relative error $err_{21}$ between the solutions. This error for any quantity $\phi$ is defined as:

$$err_{21} = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right|$$ (5.59)

where the subscripts denotes the grid refinement level. The lower value is for the finer grid and the larger is for the coarser grid. According to the procedure
in Celik et al. (2008) the apparent order $o_a$ of the numerical approach can be computed as follows:

$$o_a = \frac{1}{\ln(r_{21})} |\ln(|\epsilon_{32}/\epsilon_{21}|) + g(o_a)|$$

(5.60)

$$g(o_a) = \ln\left(\frac{r_{21}^{o_a} - s}{r_{32}^{o_a} - s}\right)$$

(5.61)

$$s = 1 \cdot \text{sign}(\epsilon_{32}/\epsilon_{21})$$

(5.62)

where $\epsilon_{ij}$ is the difference in $\phi$ between grids $i$ and $j$. Since a grid refinement factor $r_{ij}$ of 2 is adopted between grids it can be seen that $g(o_a)$ is zero. A solution on a hypothetical infinite resolved grid can be computed by means of extrapolation:

$$\phi_{ext} = \frac{r_{21}^{o_a} \phi_1 - \phi_2}{r_{21}^{o_a} - 1}$$

(5.63)

which is shown with a thick gray line in Fig. 5.5. This line is seen to be close with the Fine grid solution. Moreover, the relative error between the fine grid level solution and the extrapolated result can be computed as follows:

$$\text{err}_{ext} = \frac{\phi_{ext} - \phi_1}{\phi_{ext}}$$

(5.64)

From Celik et al. (2008) the Grid Convergence Index (GCI) is defined as follows:

$$GCI = \frac{1.25 \text{err}_{21}}{r_{21}^{o_a} - 1}$$

(5.65)

which can be used to estimate the uncertainty of the extrapolated result. Results for the GCI and the relative errors are listed in Table 3.

<table>
<thead>
<tr>
<th>$r$ (mm)</th>
<th>$\text{err}_{ij}$</th>
<th>$\text{err}_{ext}$</th>
<th>GCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>55.5</td>
<td>1.7%</td>
<td>0.5%</td>
<td>0.7%</td>
</tr>
<tr>
<td>72.5</td>
<td>0.7%</td>
<td>0.07%</td>
<td>0.09%</td>
</tr>
</tbody>
</table>

Table 3: Listing of grid convergence results based on static pressure in the diffuser at two different radii.

The analysis is complemented by investigating the sensitivity of Mach number and pressure coefficient to the grid resolution, respectively. In Fig. 5.6 these quantities are circumferentially averaged on the $m-\theta$ coordinate system from a station one inducer diameter upstream of the impeller to the diffuser exit, see Fig. 5.2 for orientation purposes.

Both the Mach number and the pressure coefficient show a characteristic rise through the impeller. Their distributions are relatively insensitive to
5.2. ASSESSMENT OF ACCURACY

Figure 5.6: Mach number and pressure coefficient on the $m - \theta$ coordinate system from a station one inducer diameter upstream of the impeller to the diffuser exit for different grids. The extrapolated solution for an infinite grid is included with the GCI uncertainty indicated with errorbars. The operating condition is near optimum efficiency at $\omega = 64000$ rpm.

the grid resolution. Further refinement has therefore a marginal effect in prediction of global performance quantities such as the total pressure rise in the compressor stage.

While assessment of the grid dependency in terms of statistical data is common it is also relevant to analyze the range in which the fluctuating flow structures are represented by the chosen grid resolution. For this purpose the Power Spectral Density (PSD) of pressure history signals in two different probe points are computed with a time Fourier transform. The PSD describes the total signal power over the frequency domain and is analyzed for all three grids as shown in Fig.5.7. The mid frequency content is captured by all grids in the diffuser probe where the average cell edge length is according to Table 2. Downstream of this point at $P_{out}$ the grid is gradually stretched towards the outlet of the computational domain which therefore results in reduced resolution. At the higher frequency range one can observe that with successive grid refinement the energy decay approaches the -7/3 slope. It may also be observed that the signal is broadband in both mid and high frequency range and the characteristic blade passing frequency (BPF) narrowband tonality with higher harmonics are capture by all grids. Since the spectra is evaluated for only 20 revolutions of the impeller the signal sample length is too short to
Figure 5.7: Power Spectral Density of pressure for the three grids, see Fig. 5.1 for probe point locations. Probe D0 to the left and probe $P_{out1}$ to the right.

resolve the low frequency range, which is therefore truncated in the figures. The cutoff frequency for the coarse grid is clearly severe but acceptable for the finest grid level considered since both the BPF tonality and broadband content are consistent with the experimental data. Overall, the result from the spectra indicates that the adopted implicit LES approach do not introduce artificial amplification at high frequencies. Hence, the use of the inherent numerical dissipation of the adopted discretization schemes to represent the dissipative nature of the smallest turbulent eddy scales is demonstrated.

To capture relevant features in the low frequency range, a longer time history is needed. This has been performed with the fine grid. Figure 5.8 shows the PSD in two different probe points for the LES data, and compared with the experimental data. It can be observed that with a sufficiently long run time, a broad surge peak including the first surge harmonic is captured. Moreover, interesting narrowband features are also captured at 0.5RO. In probe point $IS_{1}$, narrowband features are captured at RO, 2RO and 4RO. Those features are less visible in probe $D_{0}$.

Stereoscopic Particle Imaging Velocimetry (SPIV) measurements of this centrifugal compressor with a bell-mouth intake has been done by orienting two cameras to look on a laser sheet, see works by Guillou et al. (2012, 2010). The SPIV set-up considered different orientations of the laser sheet and cameras, so that velocity data in a mid-longitudinal side-view plane section as well as in some cross plane sections upstream of the impeller face were acquired. This offers the opportunity to compare the computed LES velocity flow field against SPIV, as seen in Fig. 5.9 and Fig. 5.10. The figures show the time-averaged velocity magnitude contours obtained by SPIV measurements and
5.2. ASSESSMENT OF ACCURACY

Figure 5.8: Power Spectral Density of pressure at probe point locations $D_0$ and $IS_1$.

LES simulation for the finest grid with 1 degree impeller rotation per time-step for Design condition at 64000 rpm.

Figure 5.9: Time averaged velocity magnitude contours obtained by SPIV measurement and LES simulation on the side plane section for Design condition at 64000 rpm.

The SPIV visualization shown in Fig. 5.10 exhibits some local arch shaped structures in the compressor inlet region. Since the cameras are oriented with an angle onto the laser sheet, the observed arch shapes are due to the asperity contours on the ported shroud wall being reflected in the background. Overall,
5. CASES AND RESULTS

Figure 5.10: Time averaged velocity magnitude from SPIV measurement on plane section P2 compared with LES data for Design condition at 64000 rpm.

There is a fair agreement between the LES data and the SPIV measurements in terms of the velocity magnitude levels. In Fig. 5.10, it can be observed that the velocity magnitudes are slightly higher at the supported ribs of the ported shroud (see Fig. 5.2 for locations of the ribs). These velocities evolve in the jet-like structures described by Jyothishkumar et al. (2013). The highest velocities are also measured at these locations in the SPIV measurements.

The relative contribution from pressure and wall shear stress forces respectively on torque and power are assessed. In Table 4, the computed torque of the impeller wheel is given.

<table>
<thead>
<tr>
<th>Part</th>
<th>Torque from pressure</th>
<th>Torque from wall shear stress</th>
<th>Net torque</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller</td>
<td>2.19 Nm</td>
<td>3e-2 Nm</td>
<td>2.2 Nm</td>
</tr>
</tbody>
</table>

Table 4: Torque calculation on the impeller wheel.

The contribution from wall shear stress is less than 1% of the net torque. This suggests that viscous effects near the wall are two orders of magnitude smaller than pressure forces. Even though a modeling error due to the wall functions employed might lead to an error, the effect on the global performance is small. The power is given as follows:

\[
\text{Power} = \text{Net torque} \cdot \omega \approx 15 \text{ kW} \quad (5.66)
\]

where \( \omega \) is the rotation speed.

5.3. Steady-state flow calculations

The flowfield characteristics of the small and large compressors shown in Fig. 5.1 are given in Fig. 5.11 for the same impeller speed-line (normalized \( u_{redC} = 19 \)).
They are designed to operate in different mass flow rate regimes. The smaller compressor is aimed for the usage at lower mass flow rates, while the larger compressor can cope with larger mass flow rates for e.g. a higher performance engine. The flow enters from the left into the domain and is accelerated towards the impeller. The compressor wheel is turning into the clockwise direction, when observed from the inlet. At the tips of the wheel, the highest flow velocities are provoked. The imposed kinetic energy by the wheel blades is converted into thermal energy at high stagnation pressure in the diffuser (the flow velocities decrease). From the diffuser, the stream exit tangentially into the volute. Since the volute increases its cross-sectional area in clockwise direction (consistent with the wheel rotation), the pressure increases towards the outlet, see Fig. 5.11. The observed flow features for the two compressor designs are similar at the near optimum efficiency operating condition.

Towards the surge line, lower momentum acts against a high outlet pressure. The generated kinetic energy by the impeller is lost in irreversible flow losses, with a resulting drop in stagnation pressure. Flow losses are manifested as local flow reversal. Thus, the flow is pushed from the diffuser region through the tip leakage (at the periphery of the impeller) against the main flow direction. This is obvious for both compressors in Fig. 5.11 (left column), where the annular high velocity flow can be seen at the outer periphery. This flow feature is characteristic towards the surge margin and lowers the compressor performance. The reversed flow emerging from the tip leakage reaches further upstream for the larger compressor. This is an effect of the larger compressor operating at higher mass flow rates towards its surge line. The interaction between the reversed flow near the periphery of the impeller and the incoming flow triggers the surge occurrence at different locations in the compressor map.

Figure 5.12 shows the velocity vector field together with normalized Turbulent Kinetic Energy (TKE) contours around the blades near the shroud. As approaching the surge line, i.e. reduced mass flow rate, separated flow can be observed on the suction side of both the main and splitter blades. It commences on the mid-blade towards the rear of the blades. This relates with the increased incidence flow angle upstream of the blade passages. For the lowest mass flow rate considered for each compressor the upstream velocity field exhibits a spatial wave characteristics which relates with partially blocked flow. Generally, the normalized TKE levels amplify with reduced mass flow rate and perceptible levels are localized near the splitter blade leading edge. This relates to a growing velocity gradient and flow separation towards surge condition. Under such conditions this may lead to a Kelvin-Helmholtz instability with periodic vortex formation, see Bousquet et al. (2015). The overall flow field evolves generally similar for the two different compressors. Towards the diffuser inlet the flow accelerates, and then being gradually compressed to high pressure in the volute.
5. CASES AND RESULTS

Figure 5.11: Mach number and pressure coefficient contours on side and front view planes (small compressor at the top rows and large compressor at the bottom rows for speed line $u_{redC} = 19$). The pressure coefficient is obtained by normalizing the pressure with $\frac{1}{2} \rho_{ref} u_{redC}^2$, where the reference density is $\rho_{ref} = 1.25 \text{ kg/m}^3$.

Figure 5.13 compares the axial velocity profiles upstream of the impeller face for all points on the speed line $u_{redC} = 19$. Near the shroud surface towards the surge line the flow separates with flow reversal going in the upstream direction. For more stable operating conditions and towards the choke line the profiles approach similar solutions with no separation at the shroud. Closer to the compressor axis at $r/R = 0$ the flow is decelerating in front of the impeller nut. It can be seen that the large compressor has a narrower surge margin compared to the smaller compressor where flow reversal is seen for both the last stable and the second from last stable operating point respectively.

Figure 5.14 compares velocity profiles for all speed lines for the last stable operating point. Similarly the flow is seen to separate near the shroud surface,
5.4. Unsteady calculations

Figure 5.12: Velocity vectors and turbulent kinetic energy contour on a flattened surface at 85% blade span for speed line $\dot{m}_{\text{redC}} = 19$ (increasing normalized $\dot{m}_{\text{redC}}$ from left to right). The TKE scalar is normalized with the upstream axial velocity.

but gradually approach a fully attached flow profile for the two highest speed lines. This trend is similar for both compressors, which highlights that an approximate similarity solution might exist, which is independent of the compressor size provided that the profiles are scaled appropriately. However, the larger compressor is seen to operate with more flow reversal near the shroud surface which again suggests that this compressor has a narrower surge margin.

5.4. Unsteady calculations

Figure 5.15 shows the instantaneous Mach number with overlapped streamlines as well as the pressure coefficient for the ported shroud compressor. Results are shown for flow cases A and C as presented in Table 1. Case A is representative for near-surge to surge operating conditions and Case C is representative for near optimum efficiency operating conditions. In Case C, i.e. under stable operating conditions the velocity streamlines shows that the flow is entrained from the ambient via the bell mouth inlet and being directed towards the impeller. It can be observed that the streamlines on plane $P2$ contracts to the center line which is characteristic for an approximate uniform aligned flow upstream of the impeller eye. The impeller blades transfer tangential momentum and the flow
5. CASES AND RESULTS

Figure 5.13: Axial velocity profiles upstream of the impeller face for all operating points on the constant speed line $u_{redC} = 19$. The small compressor to the left and the large compressor to the right.

Figure 5.14: Axial velocity profiles upstream of the impeller face for the last stable operating point for different speed lines. The small compressor to the left and large compressor to the right.

gain a swirling component around the center axis. In the blade passage the flow accelerates to the diffuser inlet. This is followed by a deceleration in the diffuser due to the gas being compressed into the volute volume. A continuous pressure rise is visible in the radial direction into the volute. The streamlines are overall aligned with the general flow direction and attached to the blade surfaces and a strong swirl component in the volute region is clearly visible in the front view plane. Eventually the flow is directed towards the outlet via the
5.4. UNSTEADY CALCULATIONS

Case A: $\dot{m} = 0.05 \text{ kg/s, } \omega = 64 \text{ krpm}$

Case C: $\dot{m} = 0.28 \text{ kg/s, } \omega = 64 \text{ krpm}$

Figure 5.15: Instantaneous pressure coefficient and Mach number with overlapped streamlines for flow cases A and C as presented in Table 1, on the sections presented in Fig. 5.2. The pressure coefficient is obtained by normalizing the pressure with $\frac{1}{2} \rho_{ref} u_{red}^2 C$, where the reference density is $\rho_{ref} = 1.25 \text{ kg/m}^3$. The selected snapshot for Case A is close to the peak pressure ratio in the surge cycle. Therefore, the pressure level in the volute appears higher compared to Case C.

volute exit cone. Under stable operating conditions the flow is streamlined except for local flow unsteadiness in the ported shroud cavities and in the vicinity of the sharp edge of the impeller nut.

In operating conditions close to the surge line with reduced mass flow rate, the streamlines in the blade passages shows areas where the instantaneous flow do not align with the blade surfaces. This is an indication of separated flow and stalled impeller blades. Another difference compared with the design condition is that the strong swirl in the volute breaks up in secondary unsteady components, which are transported further downstream via the exit cone. Under the volute tongue at 2 o’clock the pressure field shows a developing adverse pressure gradient which is characteristic for large system flow pulsation in the compressor. Due to the low pressure region under the tongue some flow is directed towards this area and higher velocity can be observed. Also the pressure gradient in the radial direction in the diffuser is stronger as compared to the design condition. The momentum from the impeller thus needs to work against this stronger gradient. The other Cases listed in Tab. 1 are redundant and
hence omitted for characterizing flow driven instability mechanisms emerging as operation approach off-design condition.

In surge condition there is a high activity in the ported shroud cavity. Strong flow reversal (e.g. swirling) at the tip of the impeller’s blades for low mass flow rates (e.g. tip-leakage) induce secondary flow in the diffuser. The flow reversal and swirling is seen to reach far upstream of the impeller face and cause an inflection point in the axial velocity profile, see Fig. 5.16.

As the ported shroud compressor approach off-design operating condition towards the surge-line low and high pressure zones, respectively, are observed in the surface spectra, upstream of the impeller face, as shown in Fig. 5.17. They are located opposite each other and off the center axis of the compressor. The velocity field is seen to circulate around these zones and form large coherent vortex structures, which circulate in the impeller rotation direction. From the instantaneous isosurface, it can be observed that the vortices develop at the bell mouth inlet. In the sequence $P_1 - P_2$ at 0.5RO they are seen to rotate in the same direction as the impeller rotation. Downstream of plane section $P_2$ the large coherent vortex structure tends to break up into smaller structures, in the vicinity of the shear flow due to the interaction with reversed flow from the ported-shroud cavities. There, the unsteady shear layer is seen as a circumferential band in the vicinity of the ported shroud cavity. The surface spectra at RO indicate a flow field which resembles an harmonic to the co-rotating vortex pair at 0.5RO. A tentative interpretation is the possibility of multiple coexisting large coherent vortex structures forming at the bell mouth entrance as conditions approach the surge line.
5.4. UNSTEADY CALCULATIONS

Figure 5.17: Instantaneous isosurface of the pressure coefficient colored with the Mach number (to the left). The Fourier surface spectra at 0.5RO and RO (rotating order RO = 1066 Hz) of the pressure coefficient and the velocity distribution (on the orthogonal plane $P_1$ and $P_2$ upstream of the impeller face) is shown to the right. Result is for Case A.

Figure 5.18 depicts the Power Spectral Density (PSD) calculated based on the time-history of the pressure signal obtained for all five investigated cases in the monitor point $D_0$, see Fig. 5.2 for orientation purposes. In the spectrum calculated for Case A, a strong broad peak (8 surge cycles considered) appears at the surge frequency at 43 Hz. In the mid frequency range at 50% of the rotating order (RO = 1066 Hz) a narrowband feature is present for the near-surge condition as well as for the surge condition. This is related with the co-rotating vortex pair upstream of the impeller face, which circulates around the impeller axis at this frequency. For Case D, this feature is shifted to a higher frequency, due to the higher speed-line. Elsewhere the signal is broadband apart from a sharp tonality at the Blade Passing Frequency (BPF) at 10 times the RO, the impeller having 10 blades. For the design conditions calculated for the two considered speed lines, the signal is broadband in the whole frequency range, apart from the blade passing frequency tonality.

The unsteadiness of the flow can be analyzed by evaluating the instantaneous velocity field together with the pressure field on the 50% blade span section in the rotating impeller region, see Fig. 5.19.

In Case A some impeller blades shows boundary layer separation on both pressure and suction side. This elucidates a mix of attached as well as boundary layer flow separation depending on the impellers orientation relative to the volute. The velocity vectors show relatively large variation of the upstream
Figure 5.18: Power Spectral Density in monitor point D0. Case A – surge, 64 krpm, $\dot{m} = 0.05$ kg/s. Case B – near-surge, 64 krpm, $\dot{m} = 0.085$ kg/s. Case C – stable, 64 krpm, $\dot{m} = 0.28$ kg/s. Case D – near-surge, 88 krpm, $\dot{m} = 0.166$ kg/s. Case E – stable, 88 krpm, $\dot{m} = 0.28$ kg/s.

Figure 5.19: Instantaneous velocity and pressure coefficient fields projected on a flattened surface of revolution which is a projection to the $(m', \theta)$ coordinate system. The $m'$ coordinate is the arc length along the 50% blade span section (see Fig. 5.2) projected into the $(z, r)$ plane and normalized by the local radius (Drela & Youngren (2008)). Moreover the center is located at 12 o’clock and looking down on the 50% blade span section. The flow enters from the bottom and exit at the upper end of the figure.

flow angle as compared with design condition Case C. A low pressure zone is visible in the pressure coefficient aligned with the volute tongue at the diffuser
inlet. Low pressure zones are also visible upstream of the blade passages. The pressure fluctuations show a low pressure zone near 2 o’clock, which is aligned with the pressure gradient under the volute tongue as well as on the opposite side at 8 o’clock. This relates with the co-rotating vortex pair upstream of the impeller eye and subsequently may lead to a propagating disturbance. This phenomenon is periodic and manifest in a tonal narrowband feature in the frequency spectra. Looking closely at the trailing edge region of the blades (at 8 o’clock) the passages in this area are partially blocked with large varying flow angle entering the diffuser. At 2 o’clock there is more through flow. This relates with developing wake eddy structures at the trailing edge which propagate into the diffuser and correlate with the high tonality at the blade passing frequency, observed in Fig. 5.18.

The incident angle of the upstream flow entering the blade passage at design condition is approximately steady whereas in the surge condition larger variation is present. At 2 o’clock and 8 o’clock for the selected instantaneous snapshot the flow incident angle is seen to be higher, which relates to the low upstream pressure zones at these positions. This may lead to more reversed flow on neighboring blades. In other areas the flow incident angle is smaller.

Both hydrodynamic and acoustic pressure fluctuations are associated with the solution provided when solving the compressible flow governing equations. In the compressor the turbulence related fluctuations are orders of magnitude higher than the acoustic pressure fluctuations. In the hemisphere upstream from the bell mouth, outside the hydrodynamic region of the flow, the turbulence intensity is much lower. There the acoustic pressure fluctuations become distinctively visible and may be observed and quantified.

Figure 5.20 presents the pressure fluctuations emitted from the compressor and the predicted Overall Sound Pressure Levels (OASPL) for design and off-design operating conditions. The side view plane is shown. For the pressure fluctuations a narrow range is selected to visualize the sound waves propagating in the upstream direction away from the bell mouth entrance, towards the farfield. The OASPLs show relatively high values focused to the diffuser inlet due to the blade passing and interacting with the incoming flow, which manifest in a high tonality at the blade passing frequency. Towards near-surge conditions the intensity is qualitatively higher as compared to design condition. The OASPLs have a tulip-like structure exposed upstream of the impeller face, caused by the interaction between the shear layer in the vicinity of the ported shroud cavity and the vortex structures that are seen to evolve about the center axis. These are seen to oscillate at roughly 50% of the rotating order.

The intensity of the pressure fluctuations are orders of magnitude stronger in the vicinity of the impeller region and decay as moving upstream into the hemisphere towards the inlet of the computational domain. The observed waves in the hemisphere are seen to propagate upstream at the speed of sound. At
5. CASES AND RESULTS

Figure 5.20: Side view section colored with $p'$ with a narrow range to detect sound wave propagation in the near field upstream of the bell mouth as well as OASPL(dB). Results are shown for Case A, B and C at 64000 rpm.

Distances from the source exceeding a wavelength the pressure fluctuation falls off like the inverse of the square of the distance from the source, see Fig. 5.21.

Figure 5.22 shows the OASPL directivity upstream of the bell mouth in two orthogonal planes. The OASPL shows no significant difference in directivity between the side and the top view planes (Case A to C). A slight asymmetry is seen for the top view plane for Case C. This suggests a slight directivity with a possible elevated sector close to 8 o'clock, which is opposite to the volute tongue at 2 o'clock. However, this asymmetry is seen to even out at lower mass flow rates towards Case A. Larger noise levels are characteristic for the near-surge (Case B) and surge (Case A) scenarios.

The near field acoustic spectra in a point located on the R1 curve in the hemisphere and $\theta = 30$ deg from the vertical axis relative to the bell mouth entrance ($P_{in2}$) is presented in Fig. 5.23. The spectra in this point show both similarities and differences compared to the spectra in point $D0$ located in the diffuser. There, the tonality at the surge frequency can be seen to be amplified towards the surge line. The narrowband features at 50% of the rotating order and the rotating order itself can also be observed. Towards the high frequency range the sharp blade passing frequency is captured correctly. Elsewhere the
Figure 5.21: Root mean square of pressure fluctuation as function of distance from the impeller source along the center axis. The pressure in the farfield falls off like the inverse square of the distance from the source.

Figure 5.22: OASPL(dB) directivity upstream of the bell mouth in two orthogonal planes along curve R1, see the inset to the right for orientation purposes.

signal has a broadband character. Moreover, a broadband character is captured with an interesting feature in the 500-1000 Hz mid frequency range.

One complication of detecting sound close to the impeller is that the acoustical intensity is orders of magnitude lower than the intensity component from the turbulent fluctuations, and so the acoustical part is swamped in the SPL spectra for the \( D_0 \) point. It needs to be stressed that the low frequency range
5. CASES AND RESULTS

<table>
<thead>
<tr>
<th>Case C</th>
<th>Case B</th>
<th>Case A</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5RO</td>
<td>RO</td>
<td>2RO</td>
</tr>
<tr>
<td>3RO</td>
<td>4RO</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5.23: Sound Pressure Level in the hemisphere on curve R1 at $\theta = 30$ deg (side-view plane) for different flow cases, see Fig. 5.2 for orientation purposes.

lacks resolution, but can be enhance with a more generous simulation run time and hence sample length. Nevertheless, the dominant surge peak is captured.

5.4.1. Frequency Spectra

Figure 5.24 presents the root mean square Fourier surface spectra of the static pressure field ($Prms$). It is computed in the frequency range 0 to 20 kHz, which represents an overall mean $Prms$ inside the centrifugal compressor as well as on the impeller surface (Case A to C are shown). This analysis is performed in order to elucidate areas with amplified pressure fluctuation as the compressor approach off-design conditions. In design condition (Case C) only the impeller blade tips as well as the region in the vicinity of the exducer are subjected to some fluctuation of relevance. Upstream and downstream of the impeller the $Prms$ magnitude is inconsiderable. As conditions approach surge (e.g. Case A and B) the $Prms$ magnitude is seen to intensify at the impeller blade tips as well as in the exducer (i.e. close to the diffuser inlet). The reason for the high pressure fluctuations in those areas can be explained by boundary layer separation.

Figure 5.25 presents the root mean square (RMS) of the static pressure field in the centrifugal compressor at frequencies associated with observed instabilities in the PSD point spectra. This analysis is performed in order to distinguish strong pressure fluctuations at narrowband frequencies in surge operating condition (Case A).

At the surge period 43 Hz, corresponding to 4% of the rotating order, the highest values are localized to the volute exit. The pressure ratio is building up to a certain point until the flow gradually reverses and the pressure ratio drops back to lower values. This process repeats every surge period and
this phenomenon is commonly described as the emptying and refilling of the centrifugal compressor. Relatively high values are also seen at 2 o’clock in a local area under the compressor’s volute tongue. This relates with the adverse pressure gradient as observed in Fig. 5.15. On the left hand side of the diffuser passage from 6 o’clock to 12 o’clock strong fluctuations can also be seen.

The next narrowband feature at 0.25RO reveals pressure fluctuations at the leading and trailing edges of the impeller blades, which relates with flow separation. Similarly at this frequency the fluctuation under the volute tongue is also clear. At the narrowband feature at 0.5RO most of the fluctuations are localized as a circumferential band upstream of the impeller face. This corresponds to the aforementioned co-rotating vortex pair that circulates in the same direction as the impeller rotation. Some fluctuations can also be
Figure 5.25: RMS of static pressure colored on the side, front, $P_2$, and 50\% blade span sections as well as the impeller surface. Result is shown for Case A at frequencies associated with certain flow phenomena. The RMS magnitude is scaled with $\frac{1}{2} \rho_{red} C u_{red}^2$.

observed around the diffuser passage. At RO, i.e. the rotating speed of 1067 Hz, the RMS distribution is dominant towards the rear part of the impeller blades.

At the blade passing frequency (i.e. 10RO) the RMS levels are relatively high in the diffuser and the volute, i.e. just outside of the rotating impeller. This is because those regions are seeing the impeller blades passing by.

The time-averaged tangential velocity $u_\theta$ is shown on a front view plane under the volute tongue in Fig. 5.26 for Case A. The overlapped velocity vectors show a shear layer interface, where the tangential flow reverses close to the outer wall. This is one possible mechanism for the amplified pressure fluctuation seen in Fig. 5.25 at the low frequency range. Another possible explanation is that the flow is periodically separating and reattaching under the volute tongue.
5.4. Flow mode decomposition

The flow field at the surge operating condition exhibits complex flow structures, which are difficult to interpret in the space-time domain. Instead a lower order representation of the flow, by only the large flow structures, can be easier to interpret. The high frequency incoherent turbulent fluctuations are filtered from the flow field.

A total of 2160 instantaneous snapshots of the static pressure field as well as the velocity field have been used for the computation of DMD modes. The snapshot sequence corresponds to approximately two surge periods and with a sample frequency corresponding to ten degree rotation of the impeller between snapshots. The operating condition considered for this DMD analysis is the surge condition at 64000 rpm, Case A.

Figure 5.27 shows normalized DMD magnitude and growth rate for the mode shapes at frequencies that exhibits interesting narrowband flow features, as investigated in the previous Fourier surface spectra analysis. The DMD magnitude from the mean zeroth mode is used to normalize the other modes. In practice DMD analysis produce a large number of mode shapes, equaling the number of eigenvalues and hence snapshots considered. However, the ones depicted in Fig. 5.27 are amongst the more energetic mode shapes, including the 0th mode which represents the mean flow. Additionally, there are only a handful number of modes with significant magnitude and the impact reduces for higher order modes. Apart from the zeroth mode, the surge mode and the RO mode are the two most noticeable mode shapes. Some significance is present also with harmonics to the RO mode as well as 0.5RO mode. It
should be noted that the DMD result in this case produced several candidate mode shapes at similar frequencies, but all converged on the unit circle. By analyzing the growth rate of dominant mode shapes it can be seen that the RO and 2RO modes contributes to an overall positive growth rate and may hence be attributed some significance in driving the flow instability in Case A. The surge mode (e.g., the 2nd most energetic mode shape, as compared with the mean zeroth mode) as well as the 0.5RO mode both has negative growth rates. A tentative interpretation is that these modes are damped but they are sustained by energetic input from the RO and the 2RO modes, respectively. One limitation with the DMD analysis is that modes do not necessarily interact because of positive and negative growth rates. In fact the growth rate is sensitive to the sample number and number of full cycles covered. This limits the interpretability in terms of growth rate. With DMD, a linear operator is estimated. Hence, a negative or positive growth rate means that the mode decays or grows in time. Then non-linear effects may become dominant and damp or feed the mode. Turbulence is dispersive and non-linear, and can feed the bordering frequencies, while acoustic waves are non-dispersive (hence linear and don’t feed neighboring frequencies). The point is that care must be taken with the growth rates, when the mode is a broadband ensemble. This is because a change in sampling may alter the growth rate.

The first image in Figure 5.28 is the zeroth mode, which corresponds to the average pressure field. Higher order mode shapes should therefore be interpreted as the actual field when superimposed on the mean (zeroth) mode. This mode shows the characteristic pressure rise and an adverse pressure gradient is localized primarily to the impeller trailing edges in the vicinity of the diffuser passage. This is also seen in a small area localized under the volute tongue. Due to the adverse pressure gradient these areas may be subjected to pressure fluctuations and boundary layer separation. The other images shows the real
part of the DMD mode at frequencies associated with aforementioned tonalities as shown in the point spectra in the previous section Fig. 5.25.

At the surge period at approximately 43 Hz, a large magnitude is localized to the volute exit. The mode shape is identical with the result obtained previously with the Fourier surface spectra analysis and hence depicts the emptying and refilling process of the centrifugal compressor. The only difference is that the real part of the fluctuations is shown and not the rms values. High values are thus provoked at 2 o’clock under the volute tongue, which as stated previously relates with the adverse pressure gradient in Fig. 5.15. The DMD result also shows strong fluctuations on the left hand side of the diffuser passage from 6 o’clock to 12 o’clock, i.e. similar with the Fourier surface spectra analysis. These fluctuations are related with a boundary layer separation mode that can be seen in a DMD mode based on velocity at the same frequency, which will be described later. Therefore, the excitation of the surge period is believed to have its origin from the impeller training edge into the diffuser passage.
The DMD mode at 0.5RO depicts pressure fluctuations localized at the leading and trailing edges of the impeller blades. The fluctuation under the volute tongue is also seen and is similar with the result obtained with the Fourier analysis. It can also be observed that there is a fluctuation localized circumstantially around the diffuser passage. At the rotating order (i.e. frequency of 1066 Hz), a distinct coherent structure is visible in the rotating impeller region.

At the blade passing frequency strong pressure fluctuations can be seen in the ported shroud cavities which develop into axial bands up to the bell mouth inlet. There pressure waves are seen to propagate upstream into the hemisphere at the speed of sound. The pressure waves at the blade passing frequency are seen to radiate upstream into the hemisphere. After approximately two wavelengths upstream of the bell mouth the waves are numerically damped. Additionally, standing pressure waves are observed downstream in the outlet exit pipe at the RO as well as higher harmonics of the RO. This is due to the mass flow outlet boundary condition, which is reflective, and replicates the experimental measurement setup at UC (Guillou et al. Guillou et al. (2012)), where the mass flow is regulated with a lock-valve.

The surge mode reveals a pressure distribution that can be described as a pulsating source. In this mode the pressure field is seen to oscillate from high to low values with focus at the blade edges and towards the rear end of the impeller.
blades, see Fig. 5.29. Moreover the field is fixed relative to the impeller rotating reference frame. In the RO mode the field indicates a low pressure zone at the rear end of the impeller surface (with a negative sign). This remains aligned at 2 o’clock under the volute tongue. On the opposite side there is a mirror image with high pressure (with positive sign). Elsewhere, the distribution is neutral and hence acts as a nodal point for the mode shape. With a camera view locked to the impeller’s rotating reference frame, the distribution is seen to rotate in the counter clockwise direction at the RO. With respect to the rotating reference frame, the mode can be described as a spinning source with the axis aligned with the volute tongue. The next mode shape is at 1.5 times the rotating order. It can be observed that the mode shape has a similar spinning characteristic as with the RO mode, but here the fluctuations are focused at the leading blade edges and neutral towards the rear end of the impeller surface. The 2RO mode shape (e.g. 1st harmonic of the RO) is also seen to be spinning with respect to the rotating reference frame. Coupled with the excitation of the surge instability there is a combination of lower order mode shapes as well as higher order mode shapes, with presence of pulsating sources as well as combinations of spinning sources, respectively.

Figure 5.30 shows the normalized magnitude of the DMD vector field (based on the velocity vector field) colored on the 50% span impeller section. The zeroth mode corresponds to the mean velocity field. The higher order mode shapes superimposed on the mean (zeroth) mode represent the actual velocity vector field. The surge mode shape shows relatively low magnitude levels towards the rear part of the blade trailing edges on the suction side of the blades. The velocity vectors are seen to oscillate back and forth at the surge frequency, which relates with a global oscillation. The vectors are also seen to oscillate in the boundary layer with relates with a local instability with boundary layer separation. For half the period the vectors point radially outwards and then swap direction pointing upstream. It can be described as a pulsating mode and is seen to repeat every surge cycle.

At 0.5RO the DMD velocity magnitude shows elevated levels on one side of the exducer and the opposite side is at lower magnitudes. The vectors as well as the scalar magnitude are seen to rotate relative the curved section and spin around at the 0.5RO frequency. The higher RO mode shape is similar to the 0.5RO mode shape but elucidate two elevated areas on opposite sides of the exducer and with lower magnitudes in between.

Figure 5.31 complement the previous figure, with purpose to elucidate the interaction between the rotating impeller region with stationary upstream and downstream regions. Both the real and imaginary parts are given and the DMD mode shapes can be reconstructed using Eq. 3.44. The real part of the surge mode shape corresponds to phase angles of 0° and 180° deg, respectively. In the surge period, this relates the phases when the pressure starts to rise or when it starts to fall. The imaginary part is just phase shifted 90° deg, and
thus corresponds to the peak or the trough in the pressure ratio signal. By comparing the real and imaginary parts in the surge pulsating mode it can be seen that when the velocity vectors swap direction in the impeller, the vector field in the diffuser and in the volute starts to adjust. The area with elevated DMD magnitude from 8 o’clock to under the volute tongue at 2 o’clock, is seen to rotate clockwise. Upstream of the impeller eye a strong shear layer is visible on the side view plane. This can also be seen as a circumferential band on the P2 plane.

At the 0.5RO mode four zones with elevated magnitudes are observed in the volute. Since the imaginary part is shifted in the clockwise direction, the reconstructed mode shape shows that these zones rotate clockwise (i.e. in the same direction as the impeller’s rotation). In the works by Jansen (1964) vaneless diffuser rotating stall has been related to a local inversion of the radial velocity component in the diffuser without interaction with the rotating impeller. In
the DMD mode one interpretation is that the observed elevated zones are rotating in both the diffuser and the volute at sub-synchronous rotational speed. The flow continuously rearranges itself into areas with larger outflow and areas with larger reversed inflow. Overall this has been linked with a circumferential variation of the diffuser velocity, and driven by counter-rotating vortices, see experimental and theoretical work by Tsujimoto et al. (1996). However, the
DMD mode at 0.5RO also indicates four co-rotating vortex structures upstream of the impeller eye that are seen to rotate about the center axis in the clockwise direction.

The computed DMD mode shape at the RO mode does not reveal any clear coherent structures in the stationary regions surrounding the impeller. However, diffuse elevated zones are observed in the volute, which are convected in the clockwise direction (i.e. similar as observed with the 0.5RO mode shape). As indicated in Fig. 5.27 the computed growth rate can be linked with the mode’s tendency of triggering flow instabilities. Thus, the modes associated with surge and 0.5RO exhibit negative growth rate and may therefore be interpreted as stable in time with decaying amplitude. The RO mode on the other hand were seen to have a positive growth rate. Therefore, the flow instabilities induced at this frequency may be amplified and grow over time.
CHAPTER 6

Summary and Conclusions

Global compressor performance in terms of the pressure ratio and efficiency has been predicted numerically using a steady-state RANS approach for two different compressors. Acceptable trends with measurements in an idealized installation were achieved for a range of operating conditions for different speed-lines between the last stable operating points to choke conditions. The observed flow features for the large and small compressor designs are similar at the near optimum efficiency operating condition when the velocity and the pressure fields are appropriately scaled. As conditions approach the surge-line (last stable operating point) the mass flow rate is lower and separated flow is observed on the suction side of both the main and the splitter blade.

The possibilities of the flow field assessment in a ported shroud centrifugal compressor under stable and unstable operating conditions have also been investigated using the LES formulation. A grid sensitivity analysis was presented. For the assessment of the needed resolution with the LES approach, a study was carried out to determine the suitable spatial and temporal resolutions needed to capture the surge frequency adequately. Both spatial and temporal resolutions employed were found to not influence the global performance parameters significantly. The computational solutions were compared with available experimental measurements. Both numerical approaches captured the experimental trends of the global performance parameters. However, using the RANS approach at off design conditions may lead to significant differences as compared to the experimental data due to the steady-state assumption. In contrast to the RANS approach, the LES approach captured the trends in good agreement over a broader range of operating conditions. A fair agreement was found between the computational predictions and the experimental pressure measurements on the compressor’s back-plate at design conditions, with the LES data obtained on the finest grid being the closest to the experiments. Further, the spectral content of the pressure measurements has been compared to the monitored signals in the LES simulations for all investigated setups (i.e. different spatial and temporal resolutions). The mid frequency range content was captured by all grids. In the high frequency range, the fluctuations were damped on coarser grids due to increased numerical dissipation. For the finest
6. SUMMARY AND CONCLUSIONS

With the LES approach, the characteristic surge frequency was captured and a more justifiable range of operating conditions can be investigated. The surge cycle frequency manifests itself only after a sufficient number of revolutions requiring relatively long simulation times. Therefore, the LES approach is computationally expensive. However, the flow characteristics are more reliable represented and physical interpretations are possible.

Since the instantaneous pressure and velocity fields are rather complex to quantify over time, diverse post processing methods were used to enhance the understanding of the large flow structures occurring in the compressor flow during off design operating conditions. Modal flow decomposition methods, i.e. DMD, as well as Fourier surface spectra have been used to describe the flow fluctuations associated with certain phenomena. The complex interaction of the reversed flow off the shroud cavities circulating back into the impeller is manifested. Co-rotating vortices are seen to circulate upstream of the impeller face. The rotational structures affect the flow far into the diffuser region. However, they cause only weak pressure fluctuations, as shown by surface spectra of the pressure history at this frequency. Even though, the evaluated time range in the surface spectra and modal flow analysis was based on only some few surge periods, the important computed modes coincide with the expected surge period. A characteristic mode corresponding to the surge phenomena was found with DMD. The mode describes the pumping effect occurring with surge. At times, a large amount of fluid is pushed from the diffuser into the volute. At other times, the fluid recirculates in the diffuser in the rotation direction of the impeller and a reduced amount of fluid flows in the volute. The frequency spectra contour plotted on the surface indicates that a pulsating pressure gradient is responsible for the pumping seen in the DMD modes.

The sound pressure level point spectra in the acoustic field in the hemisphere contain both narrowband and broadband features. Towards off-design operating condition the intensity amplifies in the low frequency range, with tonality at the surge frequency, which is in accordance with da Silveira Brizon & Medeiros (2012). It is known that for critical pressure ratios, the available momentum imparted by the impeller may not be sufficient to overcome the back pressure in the volute and the flow reverses in the compressor. The triggering mechanism of the surge instability has been related to separated flow on the impeller surface (Mendonça et al. (2012), Semsitsch et al. (2013) and Galindo et al. (2015)). In the instantaneous flow field, boundary layer separation is observed starting at the trailing edges of the impeller blades. This is seen to alternate in between blade passages with varying incident flow angle upstream of the blades, in accordance with Mendonça et al. (2012) and Karim et al. (2013). Moreover, this manifests in the wave-like character in the overall pressure ratio. In the surface spectra amplified pressure fluctuations are
observed on the mid part of the blades down to the rear part of the blades. Additionally, towards surge conditions a reversed pressure gradient is manifested under the volute tongue at 2 o’clock. High pressure fluctuation is also observed from 5 o’clock to 12 o’clock as well as in the volute exit pipe, which is related with the so-called emptying and refilling of the compressor during the surge condition. The modal flow decomposition of the pressure field on the impeller surface exhibits a strong pulsating source at the surge frequency. The pressure field is observed to oscillate from high to low values with focus at the mid part of the blades towards the rear end of the impeller surface. Moreover, the field is seen to be fixed relative to the impeller rotating reference frame. In the modal flow decomposition, based on the velocity field at 50% blade span the axial velocity component is seen to oscillate at the mid of the blades.

Amplified broadband intensity is observed in the mid frequency range with narrowband features at 50% of the RO as well as the RO. Mendonça et al. (2012) found a similar amplified intensity in this frequency range for but for a straight pipe inlet configuration and relates this to a rotating instability. In the bell mouth configuration the narrowband feature at 50% of the RO relates to the interaction of the shear flow in the vicinity of the ported shroud cavity with the large coherent unsteady vortex structures upstream of the impeller face found in the surface spectra.

The tonality at the RO relates to a spinning pressure distribution at the rear end of the impeller surface found in the LES data together with the flow decomposition. There, a local low pressure area is observed to rotate on the impeller surface and being aligned with the low pressure area under the volute tongue at 2 o’clock. On the opposite side at 8 o’clock there is a high pressure area and a neutral anti-node elsewhere on the impeller surface. This relates with propagation of acoustic waves upstream in the hemisphere at the speed of sound. The flow decomposition based on the velocity field at the RO shows that the velocity magnitude is more elevated on one side of the exducer and is seen to rotate relative the impeller.

Higher up in the mid frequency range, a second amplified broadband intensity is observed in between two times the RO and four times the RO frequency, in agreement with Tomita et al. (2013) as well as Evans & Ward (2005) who designates this as the so-called whoosh noise. Whereas Raitor & Neise (2008) relates this to tip clearance noise due to leakage between the shroud and the blade tips. Galindo et al. (2015) found no correlation between tip clearance size and the acoustic noise generation. The SPL point spectra in the hemisphere show narrowband features in this frequency interval, with tonality at higher harmonics to the RO. This correlates with a number of harmonic mode shapes found with modal decomposition at relevant magnitudes. The 1st harmonic at two times the RO is seen to be significant in this frequency interval. In this mode two low and two high pressure nodes, respectively, are manifested and neutral elsewhere on the impeller surface.
In the frequency surface spectra strong turbulent fluctuation are observed at the blade passing frequency, which is due to the trailing edge wake eddies propagating into the volute via the diffuser. Moreover, strong fluctuations are seen in the ported shroud cavity, manifesting in pressure waves radiating upstream into the hemisphere at the speed of sound. The main findings can be summarized in the following main points:

- The low frequency mode describing the filling and emptying processes during surge were captured with LES.
- From DMD, a dominant mode were captured at surge which can be described as a system pulsation. This mode also depicts a periodic boundary layer separation on the impeller blades. The mode has a negative growth rate and is damped.
- A narrowband feature at half of the rotating order of the shaft was identified to correspond to two dominant co-rotating vortices upstream of the impeller face as well as four elevated cells propagating in the diffuser and volute at the same direction as the impeller rotation. This feature is characterized as a spinning mode shape with a negative growth rate.
- Dominant mode shapes were also found at the RO and harmonics to the RO, which depicts a spinning feature at the diffuser inlet.
CHAPTER 7

Outlook

7.1. Personal contributions

The characteristic surge frequency was identified both with respect to global and local performance parameters. The success in predicting the surge frequency by means of simulation can be attributed to the assessment of the flow solver employed.

The hemispherical shaped domain considered upstream of the compressor entrance, is a major difference as compared with previous numerical studies carried out on the ported shroud compressor (Hellstrom et al. (2010); Jyothishkumar et al. (2013); Semlitsch et al. (2013)). Close to the impeller the acoustical intensity is orders of magnitude lower than the intensity component from the turbulent fluctuations and so the acoustical part is swamped in the SPL spectra. In the hemisphere, sufficiently far upstream, the acoustic field becomes distinctively visible. Also, further disposal of advanced post processing techniques DMD and Fourier surface spectra have added another dimension to compressor surge analysis and compressor flow instabilities.

From literature survey of different centrifugal compressor flow studies, it is suggested that tip clearance noise occurs in a narrowband range of half the rotating order. This may be interpreted as a definition by the literature, i.e. the narrowband noise at the half of the rotating order is the tip clearance noise. The term "tip clearance noise" comes from axial compressors, and has been embraced in the centrifugal compressor community. It is unclear if it is defined in terms of a generation mechanism. The task would be to illustrate the generation mechanism that causes this narrowband noise. Is it related to the tip clearance or not? Most of these definitions originate from experimentalists in the 50-70ies. They did not have computers to explore the flow field, which is possible today. The coherent structures upstream of the impeller face illustrate a different feature. Perhaps it has been interpreted as rotating stall in the past. When a pressure measurement is performed it is arguably challenging to see the flow. It may therefore be interpret as rotating pressure signature for rotating stall but in reality it may be something else.
7.2. Limitations and Continuation

While a specific flow mechanism causing surge is yet to be pinpointed, the simulation result does predict existence of possible flow driven instability candidates such as boundary layer separation and shear layer instabilities that may trigger rotating stall and surge. In the quest to determine a surge inception scenario it is suggested to compare the DMD modes for several points on the same speedline as conditions are gradually stepped from near optimum efficiency towards off design conditions. In so doing, the aim would be to find a particular operating point where a precursor flow mode might be present, i.e. prior to manifestation of rotating stall and surge.

Direct sound propagation with LES is typically only computationally affordable for near field acoustic analysis. This is due to the requirement of having a sufficient number or grid points per propagated wavelength in the farfield. However, LES may be combined with a second numerical approach that handles only the acoustic wave propagation, such as the Ffowcs Williams-Hawkings model. This is suitable for compact acoustic sources, which are enclosed by a surface completely separated from the microphone locations. For many applications, it is more relevant to analyze the acoustic noise radiating in the farfield rather than characterizing the sources as such, since the observers are not located in the very proximity of the source.

The time to achieve an accurate solution with LES is several orders higher than with the relatively fast RANS approach. Running RANS simulations for assessing the flow in the centrifugal compressor near an optimal efficiency operating condition can be expedite in about three hours on a modern multi-core processor. Thus, RANS is today the workhorse method used in evaluating compressor maps. Definitely, it can produce reasonable agreement with measurements in terms of performance parameters for near optimal efficiency conditions. However, as the operating conditions approach the surge line, there is no unique steady-state solution due to the global unsteady flow oscillations. One could consider running unsteady RANS but this is in general only recommended for tonal narrowband flow features as obtained from blade passing or coherent shedding.

Parametric compressor performance prediction models are essential in early compressor design stages and for integration as part of a larger system such as a turbocharger in an internal combustion engine. These models are based on many assumptions and semi-empirical correlations in order to allow fast and affordable design parametric studies. Zero-dimensional and steady-state assumptions instead of unsteady three-dimensional flow in complex geometries are reasons why these models may fail to accurately predict compressor performance at off-design conditions. They are often validated using measured overall performance such as pressure and temperature ratios for the impeller, diffuser, or whole compressor. This is because flow field data from the internal
compressor flow is challenging to obtain experimentally. Therefore, it is difficult to separately evaluate the effect of different loss types such as incidence loss or friction loss from internal compressor flow. The comparison of these loss models with data from numerical simulation, which allows high quality flow visualization and extraction of detailed flow data, can therefore be of value in understanding the different loss types, and the accuracy of their respective models.
A number of publications have been produced in the frame-work of this Licen-
tiate work. They are listed in a chronological order as follows:

**Paper 1**  
*Assessment of the 3D Flow in a Centrifugal compressor using Steady-State and Unsteady Flow Solvers.*  

A grid dependence study of RANS and LES data were performed on a ported shroud centrifugal compressor. The characteristic surge frequency was captured with LES. POD/DMD and surface spectra post-processing were tailored for the LES data. Some flow-driven instabilities were identified and characterized. The candidate (ES) carried out all the simulations, which involved pre-processing of the CAD geometry, volume meshing, solver run, post-processing of data and analysis, and wrote the paper. The work was supervised by MM. The experimental data were provided by Prof. Gutmark at University of Cincinnati. MM & BS provided feedback concerning the entire work and commented the manuscript. The paper was presented by ES at the SAE 2014 International Powertrain, Fuels & Lubricants Meeting, Birmingham UK 2014.

**Paper 2**  
*Similarities and Differences Concerning Flow Characteristics in Centrifugal Compressors of Different Sizes.*  

Compressor maps were produced using a RANS approach on coarse grids for two centrifugal compressors of different sizes. Results were found in an overall good agreement with available experimental data. Similarities were observed when flow (e.g. velocity and pressure) were appropriately scaled. ES performed all CFD work involving pre-processing, solving, post-processing and analysis.
Flow-acoustics coupling in a ported shroud centrifugal compressor were assessed with LES. Wave generating mechanisms were captured in the rotating impeller (e.g. pulsating and spinning mode shapes). These were related to narrowband features in Fourier point spectra based on pressure. Propagation of mode shapes were followed and observed in the upstream acoustic near field. A characteristics loud surge noise were captured in the sound pressure level point spectra and found to correlate with experimental data from other studies. The work was performed by ES under supervision by MM. The candidate (ES) carried out all the simulations (i.e. from setting up the case, running the computations, post-processing and analyzing the data) and wrote the paper. The CAD geometry and experimental data were provided by Prof. Gutmark at University of Cincinnati. Both MM & BS provided feedback concerning the work and commented the manuscript. The paper was presented at the 21st AIAA/CEAS Aeroacoustics Conference, Dallas, Texas, USA 2015 by MM.
Acknowledgements

I would like to thank my main supervisors Prof. Mihai Mihăescu and Prof. Laszlo Fuchs for their excellent support, guidance and passion in exploring the unknowns and pushing the boundaries in the research community. It is a great pleasure participating in the Monday seminars and discuss findings with everyone in the team. A big thanks to everyone in the lab for the great environment as well as everyone at Osquars backe. It is a privilege to collaborate with so many talented researchers.

A special thanks goes to my family for all the encouragement and being an inspiration, Jan, Sunniva and Ellinor. I’m truly indebted to you for always believing and supporting me. Great thanks to all my friends for enriching my life with activities between intense research periods. To all former colleges at Saab Bofors Dynamics and CD-adapco for the great many challenges.

This work was carried out within the framework of the Competence Center for Gas Exchange (CCGEx). The support from the Swedish Energy Agency (STEM), Volvo Cars, Volvo GTT, and SCANIA are greatly appreciated. The project was also sponsored by BorgWarner who provided CAD and experimental data for the small and large compressors, respectively. The Swedish National Infrastructure for Computing (SNIC) via HPC2N and PDC, the Parallel Computing Center at KTH, are acknowledged for providing the necessary computing resources. I’m grateful to Prof. Ephraim J. Gutmark and his team at University of Cincinnati for providing the ported shroud compressor CAD as well as experimental data which have been valuable for validating the numerical results.
References


REFERENCES


REFERENCES


REFERENCES


STAR-CCM+. 2015 CD-adapco.


