Flow instabilities in centrifugal compressors at low mass flow rate

by

Elias Sundström

December 2017
Technical Reports
Royal Institute of Technology
Department of Mechanics
SE-100 44 Stockholm, Sweden

TRITA-MEK Technical report 2017:12
ISSN 0348-467X
ISRN KTH/MEK/TR-17/12-SE
ISBN 978-91-7729-555-6

©Elias Sundström 2017
Universitetsservice US–AB, Stockholm 2017
To Ellinor, Sunniva and Jan, whose sacrifices made it possible.
Flow instabilities in centrifugal compressors at low mass flow rate

Elias Sundström
The Competence Center for Gas Exchange (CCGEx), KTH Mechanics, Royal Institute of Technology, SE-100 44 Stockholm, Sweden

Abstract
A centrifugal compressor is a mechanical machine with purpose to convert kinetic energy from a rotating impeller wheel into the fluid medium by compressing it. One application involves supplying boost air pressure to downsized internal combustion engines (ICE). This allows, for a given combustion chamber volume, more oxygen to the combustion process, which is key for an elevated energetic efficiency and reducing emissions. However, the centrifugal compressor is limited at off-design operating conditions by the inception of flow instabilities causing rotating stall and/or surge. These instabilities appear at low flow rates and typically leads to large vibrations and stress levels. Such instabilities affect the operating life-time of the machine and are associated with significant noise levels.

The flow in centrifugal compressors is complex due to the presence of a wide range of temporal- and spatial-scales and flow instabilities. The success from converting basic technology into a working design depends on understanding the flow instabilities at off-design operating conditions, which limit significantly the performance of the compressor. Therefore, the thesis aims to elucidate the underlying flow mechanisms leading to rotating stall and/or surge by means of numerical analysis. Such knowledge may allow improved centrifugal compressor designs enabling them to operate more silent over a broader operating range.

Centrifugal compressors may have complex shapes with a rotating part that generate turbulent flow separation, shear-layers and wakes. These flow features must be assessed if one wants to understand the interactions among the flow structures at different locations within the compressor. For high fidelity prediction of the complex flow field, the Large Eddy Simulation (LES) approach is employed, which enables capturing relevant flow-driven instabilities under off-design conditions. The LES solution sensitivity to the grid resolution used and to the time-step employed has been assessed. Available experimental data in terms of compressor performance parameters, time-averaged velocity, pressure data (time-averaged and spectra) were used for validation purposes. LES produces a substantial amount of temporal and spatial flow data. This necessitates efficient post-processing and introduction of statistical averaging in order to extract useful information from the instantaneous chaotic data. In the thesis, flow mode decomposition techniques and statistical methods, such as Fourier spectra analysis, Dynamic Mode Decomposition (DMD), Proper Orthogonal Decomposition (POD) and two-point correlations, respectively, are employed. These methods allow quantifying large coherent flow structures at
frequencies of interest. Among the main findings a dominant mode was found associated with surge, which is categorized as a filling and emptying process of the system as a whole. The computed LES data suggest that it is caused by substantial periodic oscillation of the impeller blade incidence flow angle leading to complete system flow reversal. The rotating stall flow mode occurring prior to surge and co-existing with it, was also captured. It shows rotating flow features upstream of the impeller as well as in the diffuser.

Key words: Centrifugal Compressor, flow instabilities, rotational flows, rotating stall, surge, compressible Large Eddy Simulation.
Flödesinstabilitet i centrifugalkompressor vid lågt mass-flöde

Elias Sundström
The Competence Center for Gas Exchange (CCGEx), KTH Mekanik, Kungliga Tekniska Högskolan, SE-100 44 Stockholm, Sverige

Sammanfattning
En centrifugalkompressor är en mekanisk maskin där syftet är att omvandla kinetisk energi från ett roterande pumpjul genom komprimering av ett flödesmedium. En applikation innebär att öka lufttrycket i små förbränningsmotorer (ICE). Detta medger, för en given volym i förbränningskammaren, mer syre till förbränningsprocessen, vilket är en nyckel till en förbättrad energieffektivitet och minskning av utsläpp. Centrifugalkompressorer är emellertid begränsade vid icke optimala driftsförhållanden p.g.a. flödesinstabiliteter som orsakar roterande stall och eller surge. Dessa instabiliteter uppstår vid låga flöden och leder vanligtvis till stora vibrationer och stressnivåer. Sådana instabiliteter påverkar kompressorns operativa livstid och förknippas med signifikanta ljudnivåer.

Flödet i centrifugalkompressorer är komplext på grund av ett brent intervall av temporala och spatiala skalar samt flödesinstabiliteter. Förmågan att konvertera grundläggande teknik till en fungerande design beror på en förståelse av flödesinstabiliteter vid icke-optimala driftsförhållanden som begränsar kompressorns prestanda. Ambitionen med avhandlingen är att med hjälp av numerisk analys belysa underliggande flödesmekanismser som leder till roterande stall och eller surge. Sådana kunskaper kan möjliggöra förbättringar i centrifugalkompressorns konstruktion som gör att turboaggregat opererar tystare över ett bredare arbetsområde.

Centrifugalkompressorer kan ha en komplex utformning med roterande delar som genererar turbulenta flödesseparationer, skjuvskikt och vakar. Dessa flödesfenomen bör utvärderas om man vill förstå interaktionerna mellan flödesstrukturer på olika spatials områden i kompressorn. För hög noggrannhet vid uppskattning av komplexa flödesfält används Large Eddy Simulation (LES). Denna metod möjliggör upplösning av relevanta flödesdrivna instabiliteter under icke-optimala betingelser. Resultatkänsligheten med LES har undersömts med avseende på täthet i beräkningsnätet samt i storleken på tidssteget. Tillgänglig experimentell mätdata avseende kompressorns prestandaparametrar, tidsmedelvärderade hastigheter och tryckdata (tidsmedelvärderade samt spektra) har använts för valideringsändamål. LES producerar en betydande mängd temporal och spatial data. Detta kräver en effektiv efterbehandling och tillämpning av statistisk medelvärdering för att extraiera användbar information från den momentant koxiska datamängden. I avhandlingen används olika kompositionstekniker och statistiska metoder, t.ex. Fourier spekralanalys, Dynamic Mode Decomposition (DMD), Proper Orhogonal Decomposition (POD) och tvåpunktskorrelation. Dessa metoder möjliggör kvantifiering av storskaliga och

Nyckelord: Centrifugalkompressor, flödesinstabiliteter, roterande flöden, roterande stall, surge, kompressibel Large Eddy Simulation.
Preface

Flow instabilities at off-design operating conditions in centrifugal compressors are examined in the thesis. The main focus is at low flow rates and exploring the onset mechanism of rotating stall and surge. The first part of the thesis provides an overview of centrifugal compressor terminology as well as an introduction to preliminary aerodynamic design technology and analysis. This includes a brief description of reduced order modeling for fast assessment of compressor performance maps together with linearized modeling for assessment of stability limits. More advanced numerical simulation methodologies, i.e. steady-state Reynolds Averaged Navier-Stokes (RANS) and the Large Eddy Simulation (LES) approach, are also discussed. The second part of the thesis includes several papers, as listed further below. Papers 1, 3 and 5 are conference proceeding contributions whereas Papers 2, 4 and 6 have been submitted for publication. All papers have been adapted to comply with the thesis format.


December 2017, Stockholm

_Elias Sundström_
## Contents

Abstract v
Sammanfattning vii
Preface ix
Nomenclature xiv

### Part I - Overview, Outcomes and Discussions

#### Chapter 1. Introduction 1
  1.1. Background and Motivation 2
  1.2. Objectives and Research questions 4

#### Chapter 2. Theory of Centrifugal Compressors 10
  2.1. Geometry and characteristics 10
  2.2. 1D Compressor modeling theory 13
  2.3. Flow features of stall, rotating stall and surge 19
  2.4. Theoretical stability criteria 20

#### Chapter 3. Modeling Compressor Flow 29
  3.1. Reynolds Averaged Navier-Stokes (RANS) 33
  3.2. Curvature correction 35
  3.3. Large Eddy Simulation (LES) 41
  3.4. Advanced post-processing 43
  3.5. Ffowcs Williams and Hawkings equation 47

#### Chapter 4. Numerical Methods 50
  4.1. Discretization 50

#### Chapter 5. Results: Highlights and Discussion 55
Part II - Papers


Paper 2. Generation Mechanisms of Rotating Stall and Surge in Centrifugal Compressors 117

Paper 3. Centrifugal Compressor: The Sound of Surge 137

Paper 4. Acoustic signature of flow instabilities in radial compressors 163

Paper 5. Evaluation of Centrifugal Compressor Performance Models using Large Eddy Simulation Data 189

Paper 6. Analysis of vaneless diffuser stall instability in a centrifugal compressor 211
Nomenclature

\[ \dot{m} \] Mass flow rate \([\text{kg/s}]\)

\[ h \] Specific enthalpy \([\text{J/kg}]\)

\[ s \] Specific entropy \([\text{J/kg-K}]\)

\[ c \] Speed of sound \([\text{m/s}]\)

\[ f \] Frequency \([\text{Hz}]\)

\[ k \] Wave number \([1/\text{m}]\)

\[ L \] Characteristic length \([\text{m}]\)

\[ Ma \] Mach number \([-\text{]}\)

\[ p \] Pressure \([\text{Pa}]\)

\[ r, \theta, z \] Radial, Tangential and Axial coordinates, \([\text{m}]\)

\[ x, y, z \] Cartesian coordinates, \([\text{m}]\)

\[ \eta \] Isentropic efficiency

\[ \Phi \] Phase angle \([\text{radian}]\)

\[ \gamma \] Ratio of specific heat

\[ \beta \] angle \([\text{[]}\]

\[ c_p \] Specific heat at constant pressure \([\text{J/kg-K}]\)

\[ t \] Time \([\text{s}]\)

\[ T \] Temperature \([\text{Kelvin}]\)

\[ \bar{u} \] Velocity vector \([\text{m/s}]\)

\[ \bar{U} \] Mean velocity \([\text{m/s}]\)

\[ \rho \] Density \([\text{kg/m}^3]\)

\[ \omega \] Angular frequency \([\text{rad/s or 1/s}]\)

\[ \text{RO} \] Rotating Order, Normalized angular velocity

\[ \text{BPF} \] Blade Passing Frequency

\[ V \] Volume \([\text{m}^3]\)

\[ \text{Re} \] Reynolds number

\[ k \] Turbulent kinetic energy \([\text{m}^2/\text{s}^2]\)

\[ \epsilon \] Dissipation rate \([\text{m}^2/\text{s}^3]\)

Subscript

0 \hspace{1cm} \text{Total, ambient or mean variable}

1 \hspace{1cm} \text{upstream of the impeller}

2 \hspace{1cm} \text{downstream of the impeller}

\text{ref} \hspace{1cm} \text{variable is referred to reference value}

\text{b} \hspace{1cm} \text{blade}

Superscript

\text{--} \hspace{1cm} \text{Average}

\sim \hspace{1cm} \text{Favre average}

\prime \hspace{1cm} \text{Fluctuation}

Only commonly used symbols and quantities are listed above. Other abbreviations and quantities are explained in the text at first occurrence.
Part I

Overview, Outcomes and Discussions
Chapter 1

Introduction

Since the 1950s there is a variety of scientific evidence showing global warming: a climate change due to observed century-scale rise of the Earth’s average temperature. Global warming is believed to have anticipated effects such as rising sea levels, expansions of deserts, reoccurring extreme weather events and ocean acidification\(^1\). Such consequences will ultimately influence security of food supply due to decreasing crop yields and the survival of endangered species.

The human influence is concluded as the most likely cause, and one significant factor is vehicle emissions from cars and trucks on the open road. In the European Union (EU) the majority (99\%) of new registered cars and trucks rely on an internal combustion engine (ICE) as the main propulsion system\(^2\). Although in recent years, the market has seen a growing share of electrified propulsion units. However, due to the mature technology with ICE it’s projected to dominate the market for the foreseeable future. The problem is that the majority of ICE systems are based on combustion of fossil based fuels (e.g. Gasoline or Diesel), where the main combustion products are carbon dioxide (\(\text{CO}_2\)) and water. \(\text{CO}_2\) is a greenhouse gas, which contributes to the global warming. Combustion of Gasoline or Diesel may also yield toxic byproducts that are dangerous for humans and other organisms, e.g. carbon monoxide (\(\text{CO}\)), nitrogen oxides (\(\text{NO}_x\)), hydrocarbons (\(\text{HC}\)) and particulate matter (\(\text{PM}\)). The type and amount of emissions depend strongly on the local conditions (such as pressure, temperature and equivalence ratio) in the cylinder. Improving the fuel economy of ICE has a direct impact on reducing emissions, which is positive for the environment. The European Union (EU) legislation regarding vehicle emissions has become more stringent in the past few decades, and further restrictions are expected to follow \(^3\), \(^4\), \(^5\). By 2020 the objective dictates 40\% \(\text{CO}_2\) reduction for cars and light-duty vehicles and 30\% for heavy-duty trucks. Generated noise is another concern with combustion engines, which has a disturbing impact in densely populated residential areas. This is also

\(^{1}\)https://climate.nasa.gov/evidence/


\(^{4}\)https://ec.europa.eu/clima/policies/transport/vechiles/heavy/documentation.htm

regulated within EU, as a maximum sound pressure level (SPL) due to vehicle
drive-by on the open road\textsuperscript{6}.

\subsection{Background and Motivation}

ICE can be described as a reacting flow system. It is a system containing a
multi-component fluid mixture whose constituents react chemically with each
other. The mixture contains a substantial amount of chemical energy locked
within molecular bonds. Upon combustion the available chemical energy is
transformed into thermal energy, which in turn is transformed, partially, into
mechanical energy in the power-train of the vehicle, thus moving the vehicle.
For an efficient combustion process there is a need to regulate the concentration
species present in the reaction. This is so since chemical reaction of fuels like
hydrocarbons are composed of a large number of reaction steps with a wide range
of time scales. In addition, turbulence imposes its own time- and length-scales
that only partially overlap with the species time-scales. Resolving all time-
and length-scales relevant to combustion is out of reach to current knowledge
and computational power. Without going into details of stoichiometric balance
equations between reactant and product species, respectively, it has been
identified by other researchers that some few so called integral parameters are
influential in defining the overall power output from ICE. The engine power
output can be modeled (in zero dimension) with an elementary equation as
follows, see e.g. Heywood (1988):

\begin{equation}
P = \frac{1}{2} \cdot n \cdot p_{me} \cdot V_{SW} \end{equation}

The engine power is thus seen to depend mainly on the engine speed \(n\), the
brake mean effective pressure \(p_{me}\) (an indication of the engine load) and the
swept cylinder volume \(V_{SW}\). The last parameter is a measure of the engine
size. These parameters are included to yield different frictional losses. For
example, the engine speed is associated with a quadratic increase in friction
loss. It is approximately constant with respect to the engine load, and increases
linearly with engine size, see experimental works by Guzzella & Sciarretta
the engine speed would theoretically have a positive effect on frictional losses.
However, too low engine speeds may introduce side effects associated with
non-smooth operations causing high vibrational levels. The next candidate
parameter for reduced frictional losses to consider is \(V_{SW}\), i.e. the engine size.
This is one of the more promising engineering directions for improved fuel
economy and thus lowering the emission levels (e.g. CO\textsubscript{2}). Reduced engine size
is commonly attributed to downsizing of the modern reciprocating combustion
engine. Downsizing naturally leads to less thermal and frictional losses but
lowers the power output. This is compensated by adding a well matched
turbocharging unit in gas exchange circuit, as sketched in Fig. 1.1. The aim is
to convert some of the exhaust gas enthalpy into rotational kinetic energy of

\textsuperscript{6}Directive 2002/49/EC relating to the assessment and management of environmental noise.
the turbine wheel, which otherwise is being wasted, c.f. Guzzella & Sciarretta (2007); Ben-Chaim et al. (2013); Leduc et al. (2003).

The turbine wheel is spooled to high rotational speeds, and so is the compressor impeller wheel, since both are mounted on the same shaft, see Fig. 1.2. This can be seen in the sliced turbocharger hardware assembly, where internal components of respective subsystem are visually exposed.

![Image](https://www.carfinderservice.com/car-advice/how-a-turbo-works)

Figure 1.1: Sketch showing the gas exchange circuit between the ICE and the turbocharger system (adapted after an image from https://www.carfinderservice.com/car-advice/how-a-turbo-works).

The main objective with the turbocharging system is to provide boost pressure, by a dynamic transfer of the impeller shaft kinetic energy into increased amount of (compressed) air towards the engine’s cylinder. The compressed air is directed from the diffuser towards the cylinder via the volute exit pipe. Further elevation of the intake air density supplied to the engine is possible by means of installing an intercooler, see Fig. 1.1. Feeding the engine with a higher density compressed air, richer in its oxygen content, may allow a faster fuel burn rate within a given combustion chamber. Downsizing also reduces the overall weight of the power-train system.

For the outlined turbocharger concept to work efficiently, losses in the energy transfer must be minimized. Ideally, the fluid flow stream should therefore be steady with marginal fluctuation disturbances through the turbocharging machine. For best fuel economy the engine should be operated at low engine speeds, which means that the turbocharger in an actual driving cycle operates at
1. Introduction

Air Inlet Exhaust
intake impeller shaft scroll volute turbine wheel
bypass channel shroud diffuser oil housing

Figure 1.2: A photograph showing a partially cut-out turbocharger hardware assembly. The centrifugal compressor system is located to the left; the oil housing system is located in the middle of the view; and the turbine system to the right hand side.

low mass flow rates. However, operation of the centrifugal compressor part of the turbocharging system is subjected to a limited operating range towards low mass flow rates, where emerging flow instabilities are known to occur. Mechanisms leading to flow instabilities are stall, which may develop into rotating stall and surge. Such flow phenomena cause large vibrations and influence the lifetime of the machine.

1.2. Objectives and Research questions

The overall thesis objective is to enhance the understanding of the mechanisms and key factors leading to stall onset in centrifugal compressors. This involves quantifying emerging flow instabilities, the process of their generation and their impact on the compressor’s operating range. Of growing importance is to expose the interlink between the flow instabilities and the aerodynamically generated sound in centrifugal compressors, as well as the role of flow-acoustics coupling and its effect on the compressor stability and performance. The thesis also attempts to provide an efficient and accurate method for modeling charging system’s stability and performance. In terms of gaining physics based understanding of the driving factors and parameters governing the noise generation process, quantifications of the dominant acoustics sources are made. This involves establishment of correlations between the acoustic sources to the propagating noise in the far-field, influencing the environment.
1.2. Objectives and Research questions

In the scientific community there are several hypothesis proposed for mechanisms leading to stall, rotating stall and the more severe surge instability in centrifugal compressors. Flow driven instabilities are reported to be influenced by adverse pressure gradients, shear-layer instabilities, boundary layer separation, and wake effects, see e.g. (Paduano et al. 2001; Hellström et al. 2012; Guillou et al. 2012; Lennemann & Howard 1968; Despres et al. 2013). The geometry is typically of a confined space nature, with complex high curvature features, see Fig. 1.2. The impeller wheel is rotating at high angular velocities (i.e. more than 100,000 rpm). The compressible flow is subjected to a wide range of spatial length scales, e.g. installation duct piping an order of magnitude larger than the impeller blade tip width. Similarly, it is also subjected to a large range of temporal scales, from the low frequency surge phenomenon at dozens of Hz to the high frequency tonality due to the impeller blade passing at tens of kHz. Concerning noise production from centrifugal compressors, low frequency tonality features of the flow associated with rotating stall and surge would have a specific acoustic sound source signature yielding a distinct interference pattern of the acoustic sound in the far-field.

A tentative precursor mechanism for rotating stall and surge is that due to boundary layer separation on the impeller blade surfaces. It is associated with frequent alteration of the incidence flow angles at the inducer and the exducer side of the impeller (Paduano et al. 2001; Tan et al. 2010). This may have profound effects on the compressors efficiency and performance and thus the impeller’s ability to produce downstream momentum and boost pressure (Bousquet et al. 2014). In scenarios where the thin boundary layer is subjected to flow reversal pressure equalization may subsequently take place. This may cause increased flow blockage in the blade passages. A high enough critical pressure in the blade passage implies that the blades can no longer produce sufficient momentum to push against the high pressure downstream in the diffuser. Therefore, the pressure ratio of the centrifugal stage may drop momentarily. It is conjectured that an impeller blade may be subjected to stall on either the pressure or the suction side (Kämmer & Rautenberg 1982; Oakes et al. 2002). If manifested on the suction side it is commonly referred to as a positive stall, whereas a negative stall is said to occur on the pressure side. With analogue to airfoils, a negative stall naturally may be less common as compared to the case with a positive stall. From an impeller blade design perspective, higher curvature is more frequent on the suction side, thus boundary layer separation is more likely taking place there. Due to the impeller rotation, the flow field is typically non-uniform over time, giving rise to circumferential varying disturbance. For example, for an impeller with clockwise rotation, the left blade may experience higher incidence flow angle and thus be subjected to more stall. In contrast, the neighboring blade to the right may experience a lower incidence flow angle, and thus a degree of lesser stall. Such non-uniformity in the blade incidence flow angle may gradually start to propagate from blade to blade. Ultimately, it may manifest in a local flow instability phenomenon termed as rotating stall. Conceivably, one or several blade passages may be
subjected to stall. Sometimes this evolving process is referred to as stalled cells. Such stall cells may also impact the flow field upstream at the inducer but also downstream in the diffuser. The number of such rotating stall cells does tend to depend on the operating condition as well as the chosen geometrical design of the centrifugal compressor (Broatch et al. 2016; Sundström et al. 2017). Nevertheless, an indicative parameter have been proposed, stating that the convection speed of the propagating stall cell should be about 50% of the impeller angular speed Jansen (1964). This figure has been observed both numerically and experimentally for a range of centrifugal compressor designs (Tsujimoto et al. 1996; Ljevar et al. 2006; Kalinkevych & Shcherbakov 2013).

There exists an alternative explanation for development into rotating stall for centrifugal compressors fitted with a vaned diffuser, as proposed in the works by (Bousquet et al. 2014). Based on Reynolds Averaged Navier-Stokes (RANS) computations, the onset of rotating stall is discussed as a boundary layer separation on the diffuser vane pressure side. This rotating stall is said to be transferred to the suction side. A pressure wave was argued to cause periodic alteration of favorable and adverse pressure gradients, leading to a sequence of separation and reattachment of the boundary layer in the diffuser passage. Additionally, a hub to suction side corner separation is a possible inception site of surge instability. However, there exist centrifugal compressors fitted with vaneless diffusers, which can also be subjected to rotating stall and surge. Surge is commonly characterized as a system instability with complete flow reversal as documented by Fink et al. (1991). This instability is accompanied by large oscillations of the mass flow rate together with high amplitude low frequency pulsation of the total pressure ratio. One reported feature with compressor surge is existence of swirling tip-leakage back-flow in the same direction as the impeller rotation with a low frequency tonality associated with an emptying and refilling process between the outlet pipe duct and the impeller entrance, see e.g. Guillou et al. (2012). The tonality of surge oscillation can be estimated using the Helmholtz resonator theory on simplified model geometry. It has been shown to give a surge frequency in the order of ten percent of the rotational frequency of the impeller shaft, (c.f. (Rothe & Runstadler 1978)). The surge frequency estimate is also seen to depend on the downstream duct volume. A possible onset route for the surge instability has also been hypothesized to be due to formation of a shear-layer in the volute, associated with a velocity gradient (Hellström et al. 2012). Such shear-layer is conjectured to be periodic and believed to reside under the volute tongue.

Some centrifugal compressor designs may be fitted with a ported shroud, which are known to induce secondary flow instabilities, see works by Guillou et al. (2012); Gancedo et al. (2016). Analysis of PIV measurements exposed possible interaction of flow reversal from the ported shroud cavities with the incoming flow. The unsteady interaction between the incoming flow towards the impeller and the reversed annular swirling flow present at unstable operating
conditions was further investigated in the present work using compressible LES data, work exposed in Papers 1 to 4.

There can be several acoustic noise generation mechanisms that can be provoked in a centrifugal compressor. The corresponding acoustic sources can be categorized according to their acoustic and appearance characteristics, commonly referred to as monopoles, dipoles and quadrupoles, see e.g. (da Silveira Brizon & Medeiros 2012). A monopole acoustic source represents the acoustic contribution from the displacement of the fluid caused by motion of a surface (e.g. impeller blade surface). A dipole source corresponds to pressure fluctuations (i.e. unsteady pressure loads) on a solid surface. For a stationary surface the dipole source may be due to unsteady flow separation, unsteady vortex shedding or vortices interacting with the surface. For impeller blades, the surface is moving through a non-uniform flow field or with a varying velocity. Quadrupole sources origins from free field turbulent fluctuations or by varying tangential shear stresses on the surface. The outcoming spectrum associated with such an acoustic source has a broadband character. A prevailing noise source with approximately uniform inflow conditions is the blade passing frequency (BPF) tonality, i.e. the impeller angular velocity times the number of the impeller blades.

Narrowband tip clearance noise is discussed to overshadow the BPF noise, see Raitor & Neise (2008). It has been reported to occur at approximately 50% of the rotating order (RO) for some compressor designs and may be more substantial at lower impeller speeds. This noise is suggested to emanate from secondary flow motion in the gap between the compressor casing and the impeller blades similar to axial compressors, see works by Kameier & Neise (1997). Other researchers Mendonça et al. (2012) and Tomita et al. (2013), associate narrowband noise at higher frequencies to the rotational speed. In Mendonça et al. (2012) noise source is linked with a rotating stall feature. Another noise source observed to be evident at near-surge operating conditions is coined “whoosh noise”, see Teng & Homco (2009) or Evans & Ward (2005). It is discussed as a broadband noise, stretching over several orders of kHz. It is postulated that “whoosh noise” is more apparent at near-surge operating conditions as compared to actual deep-surge operating conditions, see Teng & Homco (2009); Broatch et al. (2015). In Karim et al. (2013) high incidence angles at the impeller blade leading edges has been found to relate with noise generation and in Broatch et al. (2015) suggests that rotating flow structures in the blade passages is causing the so called “whoosh noise”. In contrast Teng & Homco (2009) relates the main noise source to be localized further downstream in the compressor outlet piping. All these investigations reveal the challenge associated with characterizing the perceived acoustic noise with the actual generation mechanism.

In this context, gaining a physical understanding of the flow field as well as assessment of flow-acoustics coupling depends on our ability to capture,
visualize, and quantify with high-fidelity a large range of spatial- and temporal-length scales associated with particular flow and acoustic modes. A high enough resolution is desired both within the compressor, i.e. local areas with flow instabilities and acoustic sources, but also further away to understand the impact of upstream and downstream installation effects. Moreover, this should cover enough statistics to capture from the lowest (surge) up to the highest frequencies (BPF) of interest. Gaining this understanding has so far been challenging to achieve by means of available experimental tools. Nevertheless, high frequency resolved pressure probe measurements have been used to analyze narrowband and broadband flow features in the compressor, see works by Raitor & Neise (2008); Torregrosa et al. (2014). One problem is that it is an invasive technique, where drilling the compressor casing is required for probe installation. Uncertainties naturally arise if the probe tip itself may influence the flow field. Particle Image Velocimetry (PIV) is another measurement technology that has been used with some success (Guillou et al. 2012). The PIV methodology relies on taking camera snapshots of illuminated particles flowing through a laser sheet plane. As a consequence the equipment must be positioned in areas with optical access. However, if the laser sheet is positioned in the vicinity of and aligned with the impeller wheel frontal plane, some side-effects are known to occur where scattering from background solid surfaces may contaminate the image. In other words, it may highlight features of the flow which does not exist. For this reason PIV has so far had a limited practical use for a physical understanding of complex flow fields within confined space applications such, as in the case of the centrifugal compressor.

In this thesis, the Large Eddy Simulation (LES) approach is employed, which is a suitable option to extract the information needed to analyze the flow instabilities under consideration, i.e. stall, rotating stall, and surge. LES also reduces the number of modeling parameters that one encounters in for example Reynolds averaged Navier-Stokes (RANS). The modeling of near-surge conditions requires resolving the large flow pulsations along with a combination of broadband and narrowband fluctuations. Capturing of rotational flow features (e.g. rotating stall instability) in non-axisymmetric geometries calls for time-dependent and full annulus geometry (360°) simulation, which implies long computational times. Moreover, the problem involves a large range of velocities associated with the complex geometry, which is characterized by high curvatures, rotating components, tip clearances, the presence of flow control devices (e.g. ported shroud). Since LES is a discrete approximation of the flow governing equations it is believed that it can handle the complex physics (e.g. laminar-to-turbulent transition, rotating stall, surge, and noise generation) that one is expecting to encounter. An important consideration is to have a high enough spatial resolution for capturing these essential flow physics. Due to the geometrical complexity of centrifugal compressors, it is always a challenge to validate the numerical approach employed due to the limited availability of detailed experimental data in the range of parameters of interest.
Due to the chaotic nature of the flow one objective is to evaluate suitable flow mode decomposition techniques for quantifying the developing larger flow instabilities as the conditions gradually approach off-design operations. Acoustic pressure waves propagating upstream and downstream from the source regions can be correlated with phenomena in the flow and with the acoustic sources. Distribution of the acoustic source signatures may be extracted using flow mode decomposition, e.g. Fourier surface spectra, Proper Orthogonal Decomposition (POD) and or Dynamic Mode Decomposition.

High fidelity LES for these applications require sufficient computational power resources, exceeding 100000 CPU hours, i.e. supercomputing hardware facilities. Consequently, another research direction in this thesis considers the use of low order modeling for prediction of the compressor performance characteristics. This is an attractive choice since the focus of the problem under consideration can be described by few modes in addition to the fact that the approach requires low computational cost and offering fast turn-around times. Therefore, low order modeling is ideally suited for early design phases and to assess the compressors integration as part of a larger engine boosting system. The associated results have comparable accuracy as with high fidelity simulations and available experimental data at near optimum design condition. However at off-design condition towards surge or choke significant discrepancies arise with the reduced order models. In practice only the overall global performance parameters from measurements are available for correlation of the 1D models. At off-design operating conditions the LES data can be used to understand why the reduced models fail, but also to enhance their predictive capabilities (by developing augmented models for assessing component-wise parametric losses).
A centrifugal compressor can be categorized as one form of a dynamic compressor or turbomachine. The attribute “compressor” refers to compression of a fluid medium. In particular it signifies a transfer of kinetic energy to a high potential energy (pressure rise) of a continuous flowing stream. The attribute “centrifugal” states that the energy transfer involves an action due to centrifugal force. Some of the benefits with centrifugal over axial compressors are as follows: higher pressure rise may be achieved with a single stage, which also means simplicity in design and production. An axial compressor would typically need several stages in series for the same output in this regard. In general it has a better surge margin and it is less prone to foreign object damages. These properties makes them suitable for turbocharging systems of Internal Combustion Engines (ICE).

2.1. Geometry and characteristics

A centrifugal compressor can be designed in many different ways, c.f. Aungier (2000). For introductory purposes Figure 2.1 provides schematic front and side views of a typical centrifugal compressor stage. Key components are (see also Fig. 1.2): an inlet duct station (0-1), an impeller (1-2), a diffuser (2-3), a volute (4), and finally an outlet duct (5) for flow discharge where the compressed flow may be utilized for some specific purpose.

The impeller energizes the fluid by its rotation (in the clockwise direction in the figure). At the exducer station (2), the flow enters the diffuser. There, additional fluid kinetic energy is recovered before the flow discharges into the volute. In the figure the diffuser is sketched as an empty space, and is commonly referred to as a vaneless diffuser design. However, some diffuser designs may be equipped with guide vanes for flow control. Guide vanes may also be present upstream of the inducer (1), commonly denoted as inlet guide vanes. A volute is utilized to smoothly guide or collect the flow towards the exit cone and eventually being funneled out via the outlet duct. The appearance of the volute is very much based on experiences from previous well-functioning designs. A popular choice from a design perspective is to employ a cross-section area variation as function of the tangential direction. In the sketch a simple spiral function was adopted. The figure also depicts an intermediate line a little
downstream of the inducer, i.e. station \(1'\). This serves to highlight that some impeller designs incorporate blades with a shorter chord. Those blades are called splitter blades whereas the others are called full length or main blades.

A question is how and why a torque input on the impeller shaft delivers power to the fluid. It is recognized that the compressor is an open system in the sense that fluid can cross the system boundaries at the inlet and the outlet. From the first law of thermodynamics for a steady flow in such an open system there must be an energy balance. On one side of the balance equation

\[
\dot{q} + \dot{w} = \dot{m} \Delta (e + pv + C^2/2)
\]  

there is a sum of work done on the system \(\dot{w}\) and an input due to heat transfer \(\dot{q}\). The work done is due to the rigid body rotation of the impeller wall surface boundary, expressed as the angular velocity of the impeller times the applied shaft torque, \(\vec{w} \cdot \vec{T}\). The heat transfer \(\dot{q}\) is an energy flow between the system and its environment due to heating or cooling, which is most often neglected for centrifugal compressors. The equation is balanced on the right hand side with the mass flow rate \(\dot{m}\) multiplied with changes in the flow energy consisting of internal energy \(e\), work required \(pv\) to move the fluid across the system boundaries, and the kinetic energy \(C^2/2\), where \(\vec{C}\) is the fluid velocity vector. The sum \(e + pv\) in the brackets is defined as the specific enthalpy \(h\) and together with the kinetic energy term it is defined as the total enthalpy \(h_0\), respectively. Therefore, the appropriate energy equation for a compressor is

\[
\dot{w} = \dot{m} \Delta h_0 = \dot{m}(h_{0d} - h_{0i})
\]
where the energy change is taking place for a compressor operating between
inlet total conditions \((p_{0i}, T_{0i})\) and discharge \((p_{0d}, T_{0d})\). The change in total
enthalpy is often normalized with the blade tip speed \(U\) which is called the
work coefficient:

\[
\psi = \frac{\Delta h_0}{U^2}
\]

(2.3)

For incompressible flow and using \(\Delta h = \Delta p/\rho\) it can be put in the following
form:

\[
\psi = \frac{\Delta p}{(\rho U^2)}
\]

(2.4)

Instead of defining a change it is also common to define a pressure increase as a
ratio between inlet and discharge:

\[
PR = \frac{p_d}{p_i}
\]

(2.5)

For a reversible process the specific entropy \(s\) is defined as

\[
ds = \frac{dq_{rev}}{T}.
\]

(2.6)

The second law of thermodynamics dictates the direction of the compression
process from inlet to discharge. For an adiabatic process it requires \(\Delta s = 0\).
A process that is both adiabatic and reversible is said to be isentropic, i.e. a
constant entropy. For a closed system the thermodynamic relation for entropy is

\[
T ds = dh - v dp
\]

(2.7)

Thus, assuming an isentropic process, the work input required to produce the
pressure rise would be a little less than the actual \(\Delta h_0\). This is denoted as

\[
\Delta h_{0s} = \int_i^d v dp \ (s = \text{const}).
\]

(2.8)

The integrated work input between inlet and discharge is schematically drawn in
the \(h - s\) diagram Fig. 2.2. It consists of two parts, one ideal process (constant
entropy) and one isobaric process (constant total pressure). The isobaric curves
in Fig. 2.2 are obtained from the entropy change \(ds\) in the fluid:

\[
ds = dh/T - R dp/p
\]

(2.9)

In the isobaric case \(dp = 0\) and integrating yields:

\[
T_0 = T_{0ref} \cdot e^{\Delta s/c_p} \Rightarrow h_0 = h_{0ref} \cdot e^{\Delta s/c_p}
\]

(2.10)

which gives an explicit relation between the total enthalpy as function of the
change in entropy. The difference between the isobaric curves (e.g. \(p_{0ds}\) and \(p_{0d}\))
is related to the total enthalpy loss or efficiency of the centrifugal compressor
(i.e. \(\Delta h_{0s}\) compared to \(\Delta h_0\)). It is commonly defined as the fraction of the
ideal over the actual work input:

\[
\eta_s = \frac{\Delta h_{0s}}{\Delta h_0} = \frac{h_{0i} - h_{0s}}{h_{0i} - h_{0d}}
\]

(2.11)
Since most compressors operate at moderate temperatures ($< 1000$ K), a calorically perfect gas with constant heat capacities $c_v$ and $c_p$ is generally a good approximation. This allows the alternative efficiency form

$$\eta_s = \frac{PR_0^{(\gamma-1)/\gamma} - 1}{T_{od}/T_{oi} - 1},$$  \hspace{1cm} (2.12)$$

where $\gamma = c_p/c_v$ is the ratio of specific heats. In order to obtain an overview of centrifugal compressor performance for various operating conditions the pressure ratio and the efficiency quantities can be combined, see Fig. 2.2b). It is drawn with several pressure ratio (PR) curves as function of the mass flow rate, where each PR curve is given for a specific impeller speed. On top the efficiency is given as contour lines, with higher levels centered along some optimum design operating condition. At very high mass flow rates, the flow may reach sonic speeds in the impeller blade passages, and the flow becomes choked. This is a limiting boundary called stonewall or choke line. On the other side, towards low mass flow rates there is a limiting boundary called surge, which is a flow phenomenon where the compressor flow undergoes large system oscillation. Stable and robust operation is not possible beyond that limit.

Figure 2.2: Schematically drawn stage $h - s$ diagram and a compressor map. The pressure ratio is commonly based on total-to-total pressures between inlet and discharge. This corresponds to stations (0) and (5) as sketched in Fig. 2.1 (note that inlet and discharge stations may be designated with other numbers for example (1) and (2), respectively). The mass flow rate $\dot{m}$ is commonly defined at discharge. Both sketches are adopted from Aungier (2000).

### 2.2. 1D Compressor modeling theory

The compressor modeling theory formulated by Aungier (2000) and Gravdahl & Egeland (2012) is exposed in the current subsection. To relate power input
\( \dot{w} \) with the power delivered to the fluid, the velocity is considered in cylindrical coordinates, \( \vec{C} = C_r \vec{e}_r + C_\theta \vec{e}_\theta + C_z \vec{e}_z \), and the equation of angular momentum between inducer and exducer is given by:

\[
\dot{m}(\vec{r}_2 \times \vec{C}_2 - \vec{r}_1 \times \vec{C}_1) = \vec{\tau}_c
\]

(2.13)

The absolute velocity magnitude at the inducer \( C_1 \) is

\[
C_1 = \dot{m}/(\rho_{01} A_1)
\]

(2.14)

where \( \rho_{01} \) is the stagnation inlet density. For convenience the frontal area \( A_1 \) is computed at root mean square radius defined as \( r_1 = \sqrt{r_{1,\text{tip}}^2 + r_{1,\text{hub}}^2} \). The velocity vector upstream of the inducer may in practice contain a tangential component \( C_\theta_1 \) but neglecting pre-swirl allows the simplification \( C_{m1} = C_1 \), and \( \alpha_1 = 0 \), see Fig. 2.3. The impeller blade velocity is given by

\[
\vec{U} = \vec{\omega} \times \vec{r}
\]

(2.15)

where \( \omega \) is the impeller angular velocity magnitude in the clockwise direction. When \( U_1 \) and \( C_1 \) are drawn upstream of the impeller leading edge, see Fig. 2.3 the relative velocity \( W_1 \) as seen by the blade is obtained by the vector relationship

\[
\vec{W}_1 = \vec{C}_1 - \vec{U}_1.
\]

(2.16)

At the exducer the fluid exits with absolute velocity \( C_2 \), while the blade speed at this station is \( U_2 = \omega r_2 \), and the relative velocity is thus \( W_2 \). A combination of equations 2.13, 2.15, 2.16, and 2.2 subsequently gives a relation for the change in enthalpy:

\[
\Delta h_0 = U_2 C_\theta_2 - U_1 C_\theta_1.
\]

(2.17)

This equation is important and is known as the Euler turbomachinery equation. Applying the cosine theorem on the velocity triangles in Fig.2.3 one obtains:

\[
W^2 = C^2 + U^2 - 2UC \cos \alpha
\]

(2.18)

Inserting in Eq. 2.17 the Euler equation can then be written as:

\[
\Delta h_0 = \frac{1}{2} [(C_2^2 - C_1^2) + (W_1^2 - W_2^2) + (U_2^2 - U_1^2)]
\]

(2.19)

This equation shows that the work input from the rotor rotation can increase the total enthalpy of the fluid in three different ways. The first term on the right hand side \( (C_2^2 - C_1^2) \) corresponds to an increase in kinetic energy over the impeller. The second term \( (W_2^2 - W_1^2) \) is a measure of the change in kinetic energy of the relative flow. The last term \( (U_2^2 - U_1^2) \) is the centrifugal effect, which is independent of the mass flow rate and is directly connected with the inducer and exducer radii. From Fig. 2.1, \( r_2 \) is much larger than \( r_1 \) and thus the exducer blade tip velocity may approximately be twice as large as compared to the inducer blade velocity. This corresponds to a nearly 50% of the pressure rise solely due to the centrifugal effect. Since this is independent of the mass flow rate through the machine it means that radial compressors can produce a pressure rise even-though the blades may be completely stalled. In an axial compressor
the flow enters and exit at approximately the same radii. Consequently the contribution from the centrifugal term is nearly zero. Therefore, a single centrifugal compressor stage can produce much higher pressure rise as compared to a single axial compressor stage and it can do so for a wider operating range.

For an ideal impeller, the exit relative velocity should be tangential with the blade exit angle. In Fig. 2.3 the impeller blade is drawn with a back sweep angle $\beta_2 b$ and $C_{\theta,\text{ideal}}$ is given by:

$$C_{\theta,\text{ideal}} = U_2 - C_{m2} \cot(\beta_2 b)$$  \hspace{1cm} (2.20)

However, the actual tangential velocity is smaller due to a slip component $C_S$. From measurements a slip factor may be obtained to relate the fraction between the actual to the ideal case, defined as:

$$\sigma = C_{\theta 2} / C_{\theta,\text{ideal}} \approx 1 - 2/Z.$$  \hspace{1cm} (2.21)

The last expression is an explicit estimate suggested by Stanitz & Ellis (1950). A typical slip factor value range is $\sigma = 0.8...0.95$, where $Z$ is the number of impeller blades. Assuming no pre-swirl, then combining equations 2.17, 2.20,
and 2.21 yields an expression of the ideal specific enthalpy delivered to the fluid:
\[
\Delta h_{0,\text{ideal}} = \sigma U_2^2 (1 - \frac{\dot{m}}{\rho_0 A_2 U_2} \cot \beta_{2b}).
\] (2.22)
Thus, the ideal work input is a function of speed, the mass flow rate and the back sweep blade angle.

### 2.2.1. Work losses in a centrifugal compressor

It is important to note that the actual work input is smaller than the ideal situation due to various losses. Some losses are known to be more dominant and are essential in defining the stability characteristic of the centrifugal compressor. The blade at the inducer is effectively subjected to an incoming flow velocity \( W_1 \) with angle \( \beta_1 \). Depending on the mass flow rate being high or low there would be an angle of incidence, defined as the difference between the blade angle and the gas stream angle:
\[
\beta_i = \beta_{1b} - \beta_1
\] (2.23)

Hence, there must be a tangential velocity component \( W_{\theta 1} \) due to incidence that the blade leading edge is seeing. Since there is a blade the flow is effectively realigned with the blade angle. This realignment of the flow introduces a loss, which may be asymmetric depending on if \( \beta_i \) is positive or negative. The further the operation and hence mass flow rate is from an ideal optimum efficiency for the chosen blade design the larger this loss would be. If however, the loss is assumed to be symmetric about this near optimum condition is may be expressed as:
\[
\Delta h_{ii} = W_{\theta 1}^2 / 2
\] (2.24)

From the velocity triangle in Fig. 2.3 the relations,
\[
\cos \beta_1 = \frac{U_1 - C_{\theta 1}}{W_1} \quad \text{and} \quad \sin \beta_1 = \frac{C_{m 1}}{W_1},
\] (2.25)
and using the sinus relation, \( W_{\theta 1} \) is given by
\[
W_{\theta 1} = U_1 - C_{\theta 1} - C_{m 1} \cot \beta_{1b}
\] (2.26)

Therefore, the incidence loss can be expressed
\[
\Delta h_{ii} = \frac{1}{2} \left( U_1 - \frac{\dot{m} \cot \beta_{1b}}{\rho_0 A_1} \right)^2
\] (2.27)

The result shows that the incidence varies as the square of the mass flow rate and is symmetric about some optimum mass flow rate. Another vital loss is due friction. It is a typical irreversible process and introduces rise in entropy. In a proposal by Ferguson (1963), who studied friction in attached channel flows, the loss is estimated as:
\[
\Delta h_{fi} = f_i \frac{l_b}{D_i} \frac{W_{1b}^2}{2}
\] (2.28)
2.2. 1D Compressor modeling theory

There $l_i$ is the blade channel length and $D_i = 2 \pi r_1 / Z$ is the mean hydraulic diameter of the channel. The friction factor $f_i$ is often obtained experimentally but from an idea presented by Haaland (1983) an explicit formula is given as:

$$1/f_i^{1/2} = -1.8 \log(6.9/Re + [(\epsilon/D_i)/3.7]^{1.1})$$

(2.29)

It depends on the Reynolds number which defines the ratio of inertial over viscous forces in the flow and may be computed as $Re = U_2 b_2/\nu$, where $b_2$ is the impeller tip width and $\nu$ is the fluid kinematic viscosity. Combining the latter equations and using the sinus theorem, the work loss due to friction can be written

$$\Delta h_{fi} = f_i l_i / 2 D_i \rho A_1^2 \sin \beta_{1b} \dot{m} = k_{fi} \dot{m}^2$$

(2.30)

Thus, the friction loss varies as the square of the mass flow rate and is independent of the impeller speed. A similar procedure may be adopted for the diffuser friction loss:

$$\Delta h_{fd} = k_{fd} \dot{m}^2$$

(2.31)

This suggests that the diffuser friction loss also varies with the mass flow rate square. However, since the flow is not guided with vanes it is important to consider the fluid flow path and the corresponding $L$ for the friction computation. From Fig. 2.1 the width $b_2$ is constant, thus the cross-section area increases linearly with the radius. Since mass is conserved, the radial velocity must decrease. So does the tangential velocity due to conservation of angular momentum. For zero axial velocity the slope of the flow path is constant,

$$\tan \alpha_2 = \frac{C_{m2}}{C_{g2}} = \frac{dr}{rd\theta} = \text{constant},$$

(2.32)

and integrating yields

$$\theta_3 - \theta_2 = \ln \frac{r_3}{r_2} \cot \alpha_2.$$  

(2.33)

Therefore, assuming inviscid incompressible diffuser flow the fluid particles must follow along a spiral trajectory, subjected to an adverse pressure gradient. From Pythagoras theorem (see Fig. 2.3) it follows that

$$dL = \sqrt{dr^2 + (rd\theta)^2} = dr \sqrt{1 + \cot^2 \alpha_2} = \frac{dr}{\sin \alpha_2},$$

(2.34)

and integrating gives

$$r_3 - r_2 = L \sin \alpha_2$$

(2.35)

If a perfect diffusion is assumed then the average velocity along the spiral trajectory is given as $C_2/2$ and the work loss due to friction becomes:

$$\Delta h_{fd} = \frac{f_i (r_3 - r_2)}{8 D \sin \alpha_2} \frac{C_2^3}{C_{m2}^2} = k_{fd} \dot{m}^2 \left[ 1 + \left( \frac{\sigma U_2 (\rho_0 A_2)^2}{\dot{m}^2} - \cot \beta_{2b} \right)^2 \right]^{3/2}$$

(2.36)

The expression after the first equality shows a singularity in the limit where the mass flow rate approach zero, and the loss goes to infinity. Of course that is completely unphysical. In reality the loss is bounded due to viscosity. Moreover,
for shallow flow angles there would be a growing adverse pressure gradient, which may be onset for boundary layer separation and cause larger pressure losses. The expression after the second equality stipulates that as the mass flow rate escalates the fluid flow path becomes shorter, hence dropping the diffuser friction loss.

The aforementioned major losses can now be subtracted from the ideal work and the overall stage efficiency can be written

$$\eta_s = \frac{\Delta h_{0,ideal}}{\Delta h_{0,ideal} + \Delta h_{\text{loss}}},$$

where $\Delta h_{\text{loss}} = \Delta h_{ii} + \Delta h_{if} + \Delta h_{df}$. In reality more loss mechanisms should be added to this sequence. But taking into account only the major loss mechanisms, neglecting minor ones, a qualitative understanding emerge on how the stage characteristic stability is affected. The effect of the slip factor is effectively a linear shift of the work flow coefficient $\psi_s$, computed as

$$\psi_s = \eta_s \psi = \eta_s \frac{\Delta h_0}{U_2^2} = \eta_s \left( \frac{C_{m2}}{U_2} \frac{\cot \beta_{2b}}{2} \right)$$

The effects of the blade back sweep angle $\beta_{2b}$ and the diffuser friction loss results in a negative slope. This gives a stabilizing characteristic and shifts the surge line limit towards lower mass flow rates, see Fig. 2.4. Finally, the impeller loss, here mainly due to the inducer loss, gives a typical arc shape. The peak level on the $\psi_s$ curve defines the transition point from positive to negative slope. According to Greitzer (1976) surge may occur somewhere between zero and positive slope. This means that some compressor designs may surge at the peak pressure point, while some others may surge further to the left of this point. From an engineering point of view, the actual surge line is defined as the lowest mass flow rate possible to measure a stable reading from a gas stand. An extra safely margin is added to the actual surge line with aim to avoid
surge. The amount of safety margin is case specific but Bloch (2006) suggest using a few percent offset to the actual surge line. Due to lack of a universal definition, the surge line is not a reliable indicator of the ultimate transition point between stable and unstable operation. For practical considerations it is regarded as a sufficient margin to maintain stable conditions. Of course this is a very qualitative description of the stability characteristic and a fundamental question emerges if this may be quantified in practice.

Test data from gas stand measurements usually only cover the stable operating regimes, i.e. a safety margin from the surge limit on the left hand side and until the choke margin on the right hand side. This means that little is known about the characteristic in surge operation. However, it is reasonable to assume that the performance will continuously drop in the limit $\dot{m} = 0$. Crucially, due to the centrifugal effect, the compressor will still be able to produce work in this limit. Therefore, the work coefficient cannot drop to zero but must retain a positive value at $\dot{m} = 0$. For negative mass flow rates one may assume that the characteristic buildup. To mimic such behavior one may assume that the pressure rise is proportional to the mass flow rate for $\dot{m} < 0$. Introducing this idealization gives the following expression:

$$\psi_s(U_1, \dot{m}) = \frac{p_{02}}{p_{01}} = \begin{cases} c_n \dot{m}^2 + \psi_{s0}(U_1), & \dot{m} \leq 0 \\ \left(1 + \frac{\eta_s(\dot{m}, U_1) \Delta h_{0, ideal}}{\eta_s c_p T_{01}} \right)^{\gamma/\gamma-1}, & \dot{m} > 0 \end{cases} \quad (2.39)$$

where $\psi_{s0}(U_1)$ is a coefficient at $\dot{m} = 0$ and a constant $c_n$ is chosen to define the slope for negative mass flow rates.

### 2.3. Flow features of stall, rotating stall and surge

It is important to realize that the centrifugal compressor is subjected to a very complex flow field. The presence of the radial velocity component in the blade passages causes a Coriolis force/acceleration. It is normal to the relative velocity vector $\vec{W}$ and will produce a cross-flow blade-to-blade static pressure gradient. For a fluid element the Coriolis force per unit volume is

$$\vec{F}_C/V = \frac{1}{r} \frac{\partial \rho}{\partial \theta} = 2 \rho \vec{\omega} \times \vec{W} = -2 \rho \omega (W_\theta \vec{e}_r - W_r \vec{e}_\theta) \quad (2.40)$$

The negative sign indicates an increasing pressure opposite the direction of rotation, i.e. a pressure gradient towards the blade pressure side. The tangential component ($2 \omega W_r$) is usually the more influential and will direct the flow opposite to the impeller angular velocity. It is proportional to the radial velocity and will hence increase towards the exducer.

A fluid element in the blade passage is also exposed to a so called blade pressure force via the centrifugal acceleration. It has one component in the direction or tangential with the $\vec{W}$ and another component normal to $\vec{W}$. The later component acts in the opposite direction to the Coriolis acceleration. The main effect of the normal component can be seen as a force from the blade pressure side to the suction side.
Both Coriolis and blade pressure forces, respectively, can be seen to influence the fluid element in the blade passage in different ways. The Coriolis force depends on the radial velocity and will hence affect the core flow of the blade passage. The blade pressure force on the other hand only depends on geometry and impeller angular velocity and will thus mainly influence the boundary layer flow. The two forces yield a flow field developing into two separate distinct zones. Qualitatively there would be one jet like zone towards the pressure side and a more boundary layer like (slower velocity) towards the suction side. The primary jet zone is highly energetic. In many aspects it may be considered to be isentropic to first approximation. The other boundary layer like zone results in an overall secondary wake zone, and is thus considered highly non-isotropic. The two zones will gradually mix after the impeller due to different levels of tangential momentum. For compressors operating at ideal conditions the mixing length according to Johsson & Dean (1966) is approximately $1.15r_2$.

Force acting on the fluid element in the blade passage can also be due to streamline curvature. This arises from the flow being gradually redirected from axial to a radial direction. The curvature is larger on the shroud as compared to the hub side. This yields a pressure gradient directed from the hub to the shroud side. As a consequence the boundary layer on the shroud side is subjected to a stronger adverse pressure gradient, resulting in a corresponding boundary layer thickening. Therefore, the shroud is more prone to separation. On the hub side curvature will help stabilizing the boundary layer, therefore it is generally more aligned flow on that side, and less prone to separation.

Many impeller designs have unshrouded configuration, meaning that there is a gap clearance between the rotating blade tip and the stationary shroud surface. The gap is subjected to leakage with a Couette-type flow characteristic. The flow is thus leaking over from the pressure side to the suction side in the shroud boundary layer in a direction opposite the impeller angular velocity. When this flow enters the next blade passage close the suction side it may curl up into a tip clearance vortex due to boundary layer separation. The phenomenon has been studied among others by e.g. Moore & Greitzer (1986); Lennemann & Howard (1968).

2.4. Theoretical stability criteria

One of the best-known description for behavior assessment of a compressor oscillating system is the so-called Greitzer model (Greitzer 1976). The system consist of a compressor of length $L_C$ with a lumped mass on one end. The other end is connected with a throttle valve and with a plenum volume in between. This is in analogue with a mechanical system; a particle of mass $m$ who is attached to one spring and two dampers and constrained to move along a straight line, see Fig. 2.5.

From the conservation law of continuity the change in plenum mass is the difference between incoming mass (i.e. $\dot{m}_c$ at the compressor) and outgoing
mass (i.e. $\dot{m}_t$ at the throttle).

$$\frac{d \rho V_p}{dt} = \dot{m}_c - \dot{m}_t. \quad (2.41)$$

Assuming isentropic compression it can be shown that

$$\frac{d \rho}{dt} = \frac{1}{\gamma RT} \frac{dp}{dt} = \frac{1}{a^2} \frac{dp}{dt}, \quad (2.42)$$

where $a$ is the speed of sound. Therefore 2.41 can be written as

$$\frac{dp_p}{dt} = \frac{a^2}{V_p} (\dot{m}_c - \dot{m}_t). \quad (2.43)$$

The gradual flow diffusion yields low Mach number flow downstream of the compressor. Hence, assuming incompressible flow in the outlet duct the unsteady velocity variation may be described as a rigid body fluid movement due to the difference in inlet and outlet plenum pressure. It then follows that

$$\rho A_c L_c \frac{dV}{dt} = (\Delta p_c - \Delta p_p) A_c, \quad (2.44)$$

where $\Delta p_c$ is the change in compressor pressure and $\Delta p_p$ is the change in the plenum. With analogue to pipe flow, the rate of change of mass flow in the compressor can subsequently be written

$$\frac{d \dot{m}_c}{dt} = \frac{A_c}{L_c} (-\Delta p_p + \Delta p_c (\dot{m}_c, U_2)) \quad (2.45)$$

In a similar fashion the rate of change of the throttling mass flow can be expressed

$$\frac{d \dot{m}_t}{dt} = \frac{A_t}{L_t} (\Delta p_p - \Delta p_t (\dot{m}_c, U_2)) \quad (2.46)$$

The three equations 2.41, 2.45 and 2.46 together are seen to be a system of first-order ordinary differential equations. It is noted that this system is autonomous,
time \( t \) does not appear explicitly. Ideally, one should try to find an explicit solution for a given set of initial conditions, so that the system response can be assessed for any time. It is often impossible to solve a system of differential equations exactly, especially since the compressor characteristic is nonlinear with respect to the mass flow rate and impeller rotational speed. Fortunately, a step-by-step recipe for solving differential equations on a computer was devised by Euler in 1768, and more accurate recipes emerged later such as the Runge-Kutta method. At first sight, it would be necessary to employ the step-by-step program for different numerical values of the constants \( A_c, A_t, L_t, L_c, V_p, a \) and \( u_t \); not forgetting all geometrical constants involved for computing the state-space compressor characteristic. The parameter \( u_t \) is a dimensionless quantity for the throttle position (between 0 and 1). However, Greitzer deliberates the analysis, by defining the new scaled variables

\[
\phi = \frac{\dot{m}}{\rho U A}, \quad \psi = \frac{\Delta p}{\frac{1}{2} \rho U^2}, \quad \xi = t \omega_H = t a \sqrt{\frac{A_c}{V_p L_c}}
\]

(2.47)

where the first two are known as the mass flow coefficient and the pressure work coefficient, respectively. Time is here seen to be scaled with the Helmholtz frequency defined as \( \omega_H = a \sqrt{\frac{A_c}{V_p L_c}} \). Introducing this, the problem is transformed into

\[
\frac{d\phi_c}{d\xi} = B (\psi_c(\phi_c, \omega) - \psi)
\]

(2.48)

\[
\frac{d\phi_t}{d\xi} = \frac{B}{G} (\psi - \psi_t(\phi_t, u_t))
\]

(2.49)

\[
\frac{d\psi}{d\xi} = \frac{1}{B} (\phi_c - \phi_t(\psi, u_t))
\]

(2.50)

where several constant parameters are collected into two designated stability parameters \((B, G)\), defined as:

\[
B = \frac{U}{2a} \sqrt{\frac{V_p}{A_c L_c}} = \frac{U}{2 \omega_H L_c}
\]

(2.51)

\[
G = \frac{L_t A_c}{L_c A_t}
\]

(2.52)

If the throttle equivalent length is small compared to the compressor length then \( G \) would be small and the problem simplifies to a system with only two degrees of freedom \((\phi_c, \psi)\) and one stability parameter \( B \). Consider now a local linearization of the compressor performance curve and the throttle valve curve around a specific operating point \((\phi_{c0}, \psi_0, u_{t0})\) and transforming the coordinate system to the specific point yielding

\[
\begin{bmatrix}
\dot{x}_1 \\
\dot{x}_2
\end{bmatrix} = \begin{bmatrix}
B \psi'_c & -B \\
\frac{1}{B} & -\frac{1}{B \psi'_t}
\end{bmatrix} \begin{bmatrix}
x_1 \\
x_2
\end{bmatrix}
\]

(2.53)

Here \( \psi'_c \) and \( \psi'_t \) are the slopes of the centrifugal compressor performance curve as well as the throttle valve curve in the neighborhood of their respective
intersection point at \((x_1 = \phi_c - \phi_{c0}, x_2 = \psi - \psi_0, u_{t0})\). It is recognized that this is an eigenvalue problem of the general form \(\dot{x} = Ax\) with an analytical solution in the complex plane for \(\lambda(= \alpha + i\beta)\). Computing the determinant of the eigenvalue problem yields a second order characteristic equation. The two roots are

\[
\lambda_{1,2} = -\frac{1}{2} \left( \frac{1}{B\psi_t'} - B\psi_c' \right) \pm \frac{1}{2} \sqrt{\left( \frac{1}{B\psi_t'} - B\psi_c' \right)^2 - 4 \left( 1 - \frac{\psi_c'}{\psi_t'} \right)} \quad (2.54)
\]

If the real part of the eigenvalue is negative, i.e. \(\alpha < 0\), the system is stable because any perturbation will reduce in amplitude with time. Moreover, the imaginary part, \(\beta\) defines the frequency of a step response to the linearized model. Examining the eigenvalues therefore, it is seen that stability is only guaranteed for the following two conditions

\[
\psi_c' \leq \psi_t' \quad (2.55)
\]

\[
\psi_c' \leq \frac{1}{B^2 \psi_t'} \quad (2.56)
\]

The former is called the static stability condition. The second, usually more restrictive is due to the \(B\) factor, and therefore it is called the dynamic stability condition. It is interesting therefore to evaluate the root locus variation due to the stability parameter \(B\), shown in Fig. 2.6.

Figure 2.6: Root locus variation with respect to stability parameter \(B\), with choice of parameters \(\psi_c' = 1\) and \(\psi_t' = 5\).

One observes that for small \(B\) there are two statically stable equilibrium points, because there is no frequency component. As \(B\) increases, the two points move closer and closer together until they depart their separate ways. The mechanism is a bifurcation into two conjugate complex branches. At the bifurcation point then, the equilibrium position becomes dynamically stable. If \(B\) is increased some more, the branches eventually transition into the positive.
real half of the complex plane and the equilibrium becomes dynamically unstable. As $B$ continues to increase, at approximately $B = 2$, the two branches merge at a second bifurcation point and the equilibrium point is statically unstable.

This analysis contains an inherent limitation due to the employed linearization, requiring small enough variations around the equilibrium point. What is needed therefore is to maintain the nonlinear variation of the compressor performance characteristic as well as the nonlinear variation of the throttle valve curve. Experimental data obtained by Guillou et al. (2012) are presented in Fig. 2.7. Two different speedlines are depicted for this compressor, which consist of ten main blades and accommodates a ported shroud in an idealized installation with an upstream bell mouth inlet. Since it is challenging to obtain reliable measurements for compressor characteristic with positive slope, this area is approximated using a technique from Moore & Greitzer (1986). This method employs a cubic polynomial in the following form

$$\psi_c(\phi_c, N) = C_0^*(N) + H^*(N) \left[ 1 + \frac{3}{2} \left( \frac{\phi_c}{W^*(N)} - 1 \right) - \frac{1}{2} \left( \frac{\phi_c}{W^*(N)} - 1 \right)^3 \right]$$

Figure 2.7: Test data for a ported shroud centrifugal compressor (black dots). The dashed curves shows the throttle valve characteristic using Eq. 2.58. Compressor characteristic to the left hand side of the last measurement point is obtained by means of Eq. 2.57.
2.4. Theoretical stability criteria

respect to the mass flow coefficient.

\[ \phi_t = k \sqrt{\psi_t}, \text{ where } \psi_t > 0 \]  

(2.58)

In this expression \( k \) is a dimensionless throttle coefficient, which depends on the throttle position \( u_t \).

From an engineering perspective the most interesting question is what will eventually happen to the state variables \((\phi_c, \psi)\). In particular there can be four conceivable possibilities for the eventual behavior: (i) does the state variables grow exponentially with time, (ii) will the state variables tend to some finite limit, (ii) does the state variables approach some regular oscillation, (iv) or perhaps none of those things but rather fluctuate randomly. The last item (iv) can immediately be ruled out since the system considered is autonomous. However, when a nonlinearity is introduced to the system, it has the property that if nothing is done to it initially, it will remain in a state of rest for any given time. On the other hand if the stability parameter \( B \) is slightly increased, then the state variables will end up swinging back and forth in some particular way, as shown in Fig. 2.8.

In the case \( B = 0.02 \), with chosen initial conditions, \( \psi \) exhibits decaying oscillations and comes to rest at a non-dimensional time \( \xi \approx 50 \). In the phase space diagram this corresponds to an inward spiraling trajectory. For the case \( B = 0.2 \) the solution exhibits a growing amplitude and eventually approach a regular oscillation, almost similar to a simple harmonic with period 10. For still larger values of \( B \), the character of the limit cycle is gradually diverting from a simple harmonic trajectory in phase space. The period of the limit cycle is significantly reduced \( \approx 3 \) and so does the amplitude. During the first half of the cycle \( \psi \) increases slowly until the peak value. There is a sudden switch to negative \( \phi_c \) values in a very short time. During the second half of the cycle \( \psi \) reduces slowly to the valley point and then suddenly \( \phi_c \) switches to a larger positive value. Thus, the rate of the switch is directly dependent of the size of \( B \), and the larger it is the more sudden the switching becomes. The general behavior shares some resemblance with the van der Pol limit cycle, and for a large stability parameter \( B \) therefore, the sudden switch is known as a relaxation oscillation.

It is important to understand that the Greitzer model is not completely satisfactory in describing the long term behavior for a real centrifugal compressor. The shortcoming is the inability of the model to account for chaotic oscillations, since the considered system is autonomous and two-dimensional, with the following form:

\[ \frac{d\phi_c}{dt} = f(\psi, \phi_c) \]  

(2.59)

\[ \frac{d\psi}{dt} = g(\psi, \phi_c). \]  

(2.60)

The reason why the model cannot account for chaotic behavior emerges from the Poincare-Bendixon theorem (Coddington & Levinson 1955) outlined as follows. Suppose there exist equilibrium points, i.e. \( f(\psi, \phi_c) = 0 \) and \( g(\psi, \phi_c) = 0 \).
and trajectories in phase space are seeded from initial points. According to the theorem, the trajectory in phase space must either (i) terminate in an equilibrium point, or (ii) return of the seed point, (iii) or approach a limit cycle. A consequence is that for a two-dimensional and autonomous system, phase space trajectories cannot cross themselves. This prevents phase paths crossing themselves in an erratic way leading to chaotic behavior. A simple remedy then, is to consider a non-autonomous system. As the impeller blades pass by with frequency $\Omega$ one may deduce an additional pressure disturbance with amplitude $A$, being added to Eq. 2.60. This can be recast into the autonomous first order
system:

\[ \dot{\phi}_c = f(\psi, \phi_c) \]  
\[ \dot{\psi} = g(\psi, \phi_c) + A \cos \Omega \xi \]  
\[ \dot{\xi} = 1. \] 

At a stroke, the system clearly becomes three dimensional and the Poincare-Bendixon theorem is not applicable. When a small forcing is employed to the system a rather peculiar phase space emerges, as shown in Fig. 2.9. Such stochastic dynamics have two important features. First the oscillations are irregular; one period is never quite the same with the next following one. Second, the periodic behavior of the state space variable is very sensitive to initial conditions. One of the \( \psi(t) \) curves have a slightly perturbed initial condition by just 1 part in 1000. In the first few periods the solutions are almost indistinguishable but eventually, as more cycles are considered, the two curves depart, heading towards different long term behavior. The phase space trajectory given on the right hand side of Fig. 2.9 should be seen as the projection onto the

---

Figure 2.9: Numerical solution of the Greitzer model with added forcing. Initial conditions \( \psi_0 = 1.5, \phi_{c0} = 0.05 \). Choice of parameters \( A = 0.1 \) and \( B = 0.2 \) for figures on the top row. The red \( \psi(\xi) \) curve with initial condition \( \phi_0 = 1.501 \). The second row of figures display period-doubling with \( A = 2 \).
\( \psi, \phi_c \) plane, where the time dimension would be perpendicular to the plane itself. Despite appearances of phase path crossings, this never in fact happens; it is due to projection anomaly. From the seed point, annotated with a star symbol, the solution quickly spirals outward and towards the limit cycle. However, with each turn around the limit cycle the solution does not end up at the same place as with the previous cycle. The trajectory is therefore seen to wobble a little back and forth about the limit cycle in a very complex situation. Generally, this behavior may be described as a strange attractor. In the event that the solution re-occurs in phase space, thus for some later orbit, would probably require multiple turns around the strange attractor. In fact, as the forcing amplitude \( A \) is further increased, the solution display period-doubling. From the mathematical study of nonlinear systems, if a sequence of time dependent data exhibits period-doubling it is said to be a probable root to chaotic oscillations. A gradual picture thus emerges that the surge dynamics in centrifugal compressors is very complex. Even with reduced order modeling the surge feature may consist of period-doubling and possibly stochastic evolving variables. But to really understand the surge phenomenon it is necessary to study the fluid flow in more detail, which means using an unsteady, three-dimensional approach able to resolve the most important energetic flow structures.
Chapter 3

Modeling Compressor Flow

The gas flow through centrifugal compressors is assumed to be governed by the mass, momentum and energy conservation equations, respectively. Typical length scales of the flow are assumed to be orders of magnitude larger than intermolecular scales. Thus, the fluid material is treated as a continuum, meaning that flow quantities such as pressure and temperature are approximately uniform within fluid elements. The set of equations are known as the compressible Navier-Stokes equations. In Cartesian tensor notation they are given as:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \quad (3.1)
\]

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} + \rho f_i \quad (3.2)
\]

\[
\frac{\partial (\rho) e_0}{\partial t} + \frac{\partial (\rho u_i) e_0}{\partial x_i} = -\frac{\partial (p u_i)}{\partial x_i} - \frac{\partial q_i}{\partial x_i} + \frac{\partial (u_j \sigma_{ij})}{\partial x_i} \quad (3.3)
\]

Equation 3.1 governs the continuity of mass, where \( t \) – time, \( \rho \) – density, \( x_i \) – Cartesian coordinate \((i = 1, 2, 3)\), \( u_i \) – absolute fluid velocity component in direction \( x_i \). The second equation 3.2 governs the time rate of change of momentum of the fluid. It is balanced on the right hand side by a sum of normal stresses due to pressure \( p \) and shear-stresses \( \sigma_{ij} \) due to viscosity acting on the fluid. Constitutive relations must be used to connect components of the stress-tensor to the velocity gradients. The source term \( f_i \) represents the sum of body and other external forces, if such forces are present. The energy equation 3.3 is obtained via the first law of thermodynamics, where \( e_0 = e + \frac{1}{2} u_i u_j \) is the total energy per unit mass. It is balanced on the right due to heat flux \( q_i \) and stress acting on the continuum volume boundaries. Overall this constitutes five equations, but since there are nine unknown \((p, \rho, e, u_i, q_i)\) terms, additional constitutive equations are needed for a closed system. The ideal gas law is used to express density as function of temperature \( T \) and pressure \( p \):

\[
p = \rho RT \quad (3.4)
\]

\( R \) is the specific gas constant. If the fluid is a thermally perfect gas the internal specific energy \( e \) and the specific enthalpy \( h \), respectively, are only dependent on temperature. Due to compression the temperature increases, but for a centrifugal compressor application this increment in temperature is found to be
moderate (e.g. $T << 1000 \text{ K}$). Therefore, the fluid can be assume calorically perfect where specific heats (at constant volume $c_V$) and at constant pressure $c_p$) are constants. Thus, for an ideal gas

$$e = c_V T \quad (3.5)$$

$$h = c_p T \quad (3.6)$$

The enthalpy is related to energy via $h = e + RT$. Another assumption is that molecular diffusion fluxes of heat and mass may satisfy Fick’s law

$$q_i = -k \frac{\partial T}{\partial x_i} \quad (3.7)$$

where $k$ is the thermal conductivity coefficient. This yields three additional equations. Moreover, if the compressible fluid is assumed to be Newtonian, it is possible to use constitutive relations connecting the components of the viscous stress tensor to the velocity gradients:

$$\sigma_{ij} = 2\mu S_{ij} - \frac{2}{3}\mu S_{kk}\delta_{ij} \quad (3.8)$$

where $\mu$ is the molecular dynamic viscosity of the fluid. The ‘Kronecker delta’ $\delta_{ij}$ is unity when $i = j$ and zero otherwise. The first component in Eq. 3.8 depicts the deformation rate tensor due to strain and is expressed as:

$$S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \quad (3.9)$$

The second term in Eq. 3.8 on the right hand side is associated with bulk viscosity. This relation for the stress tensor assumes that Stoke’s hypothesis holds, which is commonly adopted for mono-atomic gases. The influence of the viscosity is to resist deformation caused by shear stress. Its effect is generation of a force opposite to the flow direction, where neighboring fluid elements move past each other at different flow speeds.

Since the temperature increases due to compression the dynamic viscosity is temperature dependent. This dependency can be accounted for by Sutherland’s model:

$$\frac{\mu}{\mu_0} = \left( \frac{T}{T_0} \right)^\frac{3}{2} \left( \frac{T_0 + T_S}{T + T_S} \right) \quad (3.10)$$

where $T_S$ is the Sutherland constant and $T_0 = 273 \text{ K}$ is a reference condition.

In principle, it is possible to provide any specified momentum source field configuration, for example arising from rotation in the case of centrifugal compressor applications. This could be introduced as a function involving components of the rotation vector $\omega_k$ and the radius vector $r_k$ about the impeller shaft axis.

Naturally, in centrifugal compressors there are stationary upstream and downstream components where fluid particles move in non-accelerating inertial systems. Due to the impeller rotation $\omega_k$ (assumed constant throughout this thesis), fluid particles inside this region must be treated in an accelerating non-inertial system. A kinematic relationship is used to interpret the fluid
particle velocity in the stationary reference frame $v$ (absolute velocity) to its relative velocity $v_{rel}$ in the moving (accelerating) reference frame using:

$$v = v_{sp} + v_{rel} = (v'_0 + \omega \times r_{rel}) + v_{rel}$$ (3.11)

The term $v_{sp}$ is the velocity of a material point consisting of the velocity of the origin of the moving reference frame $v'_0$ and a term due to its rotation. In the last term the angular velocity is cross multiplied with the position vector $r_{rel}$ of a fluid element moving in the accelerating reference frame. The kinematic relation is inserted into the continuity, momentum and energy conservation equations, which governs the compressible flow through the centrifugal compressor.

3.0.1. The character of the flow

In broad terms, the flow field inside the centrifugal compressor may be characterized as a highly unsteady, three-dimensional, chaotic turbulent flow. Fluctuations due to turbulence appear due to flow mixing between large and small scale flow features in a broad range of frequencies. The turbulence intensity can be estimated using the non-dimensional Reynolds number. The Reynolds number is the ratio between inertial and viscous forces. Assuming a characteristic length scale $L \approx 0.1 \text{ m}$ and assuming a kinetic viscosity of the fluid $\nu = 1.5 \times 10^{-5} \text{ m}^2/\text{s}$ the molecular diffusion time in a centrifugal compressor may be estimated:

$$T_{mol} \sim \frac{L^2}{\nu} \approx 11 \text{ min}$$ (3.12)

The turbulent diffusion time process is order of magnitude faster due to a larger viscosity of the flow. Assuming a typical eddy size say $0.5L$ and a fluctuating velocity $u'$ say 10% of the impeller blade tip speed the turbulent time scale is order of milliseconds:

$$T_{turb} \sim \frac{L^2}{\nu_{turb}} \approx \frac{L^2}{\Lambda \cdot u'} \approx 5 \text{ millisec}$$ (3.13)

The ratio between time scales of molecular diffusion and turbulent diffusion can be identified as the Reynolds number, computed as:

$$Re_\Lambda = \frac{\text{inertial forces}}{\text{viscous forces}} = \frac{\Lambda u'}{\nu} > 10^5$$ (3.14)

This indicates a highly turbulent flow regime with high diffusivity of the flow field. At the largest length scales flow eddies are limited by the characteristic geometry size and the boundary layer thickness. These eddies absorbs energy from the mean flow. A fundamental concept with fluid flow turbulence is existence of an energy cascade process. This involves an energy transfer due to interaction between large eddy scales with successively smaller ones. At the smallest scales, molecular viscosity defines a lower limit where the eddies dissipate its residual kinetic energy into heat. From the works by Kolmogorov (1991) it is conjectured that the cascaded process lead to a self-similar state at the molecular viscosity level, at which the smallest eddies would be independent
of eddies on larger scales. At the molecular level Kolmogorov obtained the following relations:

\[ \eta = \left( \frac{\nu^3}{\epsilon} \right)^{1/4}, \quad t_\eta = \left( \frac{\nu}{\epsilon} \right)^{1/2}, \quad v_\eta = (\nu \epsilon)^{1/4} \]  

(3.15)  

\[ Re_\eta = \frac{\eta \cdot v_\eta}{\nu} = 1 \]  

(3.16)

where the smallest length scale \( \eta \), time scale \( t_\eta \), and velocity \( v_\eta \) only depend on the kinetic viscosity \( \nu \) and the dissipation rate \( \epsilon \). If one assumes a power input of 10 kW and a total fluid mass of about 60 g, it gives \( \epsilon \approx 2 \cdot 10^5 m^2 s^{-3} \). Thus, the smallest length scale is about \( \eta \approx 10^{-6} m \). Another factor for the cascade process is an assumption that the energy flux from large to small eddy scales remain in a quasi-equilibrium state, \( \epsilon_f = \epsilon \). Since the ratio between large and small scales is given by:

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

(3.17)

it can be seen that the degrees of freedom needed for a 3D flow grows as \( Re_\Lambda^{9/4} \). Including the entire cascade process of all eddy scales implies that computational grid points scales as \( Re_\Lambda^{9/4} \) and the corresponding computational effort growing as \( Re^3 \). Inserting the previous Reynolds number estimate suggest a necessity of \( 3 \cdot 10^9 \) grid points. Subsequently it would be quite a challenge solving the exact governing equations of turbulent flows with all scales resolved even on the most powerful supercomputers available today.

Kolmogorov’s second hypothesis postulate on the existence of an inertial subrange, where the turbulent spectrum depends only on the wave number and the turbulent dissipation rate (and not on viscosity). For large \( Re_\Lambda \) the idea is that for a bounded range \( \eta \ll \frac{2\pi}{\kappa} \ll \Lambda \), viscosity would have a small impact. With this assumption and applying dimension analysis the following energy relations are valid in the so called inertial subrange.

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

(3.18)

The -5/3 slope has been verified in a range of high Reynolds number flow applications. A similar result but based on a pressure fluctuations is given as:

\[ E_{pp} = \alpha_p \epsilon^{4/3} \kappa^{-7/3} \]  

(3.19)

Due to the outcome of a linear regime in the inertial subrange via Kolmogorov’s second hypothesis, there is an argument to include modeling of this part of the energy spectra. In other words, instead of resolving all eddy scales all the way to Kolmogorov’s microscale, it is more affordable to employ time-averaged or filtered governing equations. This leads to a concept of turbulence modeling, which can be grouped in two categories. One is referred to as Reynolds-Averaged Navier-Stokes (RANS) with eddy viscosity turbulence modeling. The other is referred to as large scale resolving simulations, e.g. Large Eddy Simulation (LES). Figure 3.1 illustrates the conceptual coverage of the respectively modeling
approaches in the energy spectrum. Qualitatively it shows that all wavelengths are modeled with RANS whereas the most energetic flow structures are resolved with the LES approach, thus circumventing the computational challenges with Direct Numerical Simulation (DNS), which resolves all scales.

3.1. Reynolds Averaged Navier-Stokes (RANS)

Reynolds averaging is a concept where each solution quantity $\phi$ is decomposed into a time-averaged value $\bar{\phi}$ and a fluctuating component $\phi'$:

$$\phi = \bar{\phi} + \phi'$$  \hspace{1cm} (3.20)

In a compressible flow, turbulent fluctuations may cause density variations, which leads to the concept of density weighted averaging. This is also known as Favre averaging $\tilde{\phi} = \rho \bar{\phi} / \bar{\rho}$, where over-bar depicts the Reynolds average. By definition of the decomposition, the average of the fluctuations vanishes. It should be pointed out that the Reynolds averaging does not separate non-turbulent time-dependent structures from turbulent structures. By introducing the time-averaged flow quantities into Eqs. 3.1 to 3.3 yields the so called compressible Reynolds Averaged Navier-Stokes (RANS) equations:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial (\bar{\rho} \tilde{u}_i)}{\partial x_i} = 0,$$

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

and

$$\frac{\partial}{\partial x_i} \left( \bar{\rho} \left( \tilde{e} + \frac{\tilde{u}_i \tilde{u}_i}{2} + \frac{\rho u''_i u''_i}{2} \right) \right) + \frac{\partial}{\partial x_j} \left( \bar{\rho} \tilde{u}_j \left( \tilde{h} + \frac{\tilde{u}_i \tilde{u}_i}{2} \right) + \tilde{u}_j \frac{\rho u''_i u''_i}{2} \right)$$

$$= \frac{\partial}{\partial x_j} \left( \bar{u}_i (\bar{\sigma}_{ij} - \rho u''_i u''_i) \right) + \frac{\partial}{\partial x_j} \left( -\tilde{q}_j - \rho u''_j h'' + \sigma_{ij} u''_i - \rho u''_j u''_i \right)$$

\hspace{1cm} (3.23)
The viscous stress tensor $\bar{\sigma}_{ij}$ is the term responsible for transport of momentum through motions at the molecular scale. The averaging introduces the Reynolds stress term $\tau_{ij} = -\rho u_i'u_j'$, which accounts at macroscale for transport of momentum through motion at the turbulent scales. The Reynolds stress tensor is symmetric and consists of six independent components. In order to close the system or equations, for the averaged variables, the individual terms of this tensor must be expressed in terms of the averaged variables. This requirement demands introduction of modeling into the system. As stated above, in Reynolds averaging the fluctuating component includes fluctuations of any nature (turbulent and non-turbulent). If non-turbulent time-dependent structures exist and these have frequencies that overlap the turbulent spectrum, the modeling become even more difficult, if possible at all, than otherwise. In such cases, the modeling must be problem dependent and therefore it is just an ad-hoc solution.

A practical measure of the overall turbulence level is dictated by the specific turbulent kinetic energy given as:

$$K = \frac{1}{2} \left( u'^2 + v'^2 + w'^2 \right)$$  \hspace{1cm} (3.24)

Experiments and Kolmogorov’s theory indicate that fluctuations are present at all scales simultaneously. That is from the smallest Kolmogorov scales to the large length scales that are the main energy carrying eddies. The concept here with modeling is to realize the underlying mechanism associated with dissipation of energy, description of turbulence production, as well as dissipation and transport of turbulent kinetic energy.

### 3.1.1. Eddy-viscosity modeling of the Reynolds stress

The Reynolds stress tensor is commonly modeled using an idea introduced by Boussinesq. It is called Eddy-viscosity modeling where the Reynolds stress tensor is related to the time-averaged velocity gradients:

$$-\rho u_i'u_j' \approx 2\rho \nu_T S_{ij} - \frac{2}{3} \rho \nu_T S_{kk} \delta_{ij}$$ \hspace{1cm} (3.25)

where the strain rate tensor $S_{ij}$ is determined according to Eq. 3.9 and $\nu$ is the kinematic viscosity and hence a property of the fluid. The introduced turbulent eddy viscosity $\nu_T$ is a local property of the flow rather than the fluid. It must be acknowledged that several assumptions are incorporated in Eq. 3.25. One is that the Reynolds stress tensor can be obtained directly from single point quantities. Another is that the strain rate tensor assumes a linear dependence on the time-averaged velocity gradient components. Thus, there is no rotational influence. This assumption naturally limits the applicability for highly swirling flows in centrifugal compressors. Although, despite the seemingly harsh assumption, eddy viscosity modeling have been reported to give reasonable predictions much faster as compared to scale-resolving counterparts. A popular choice for eddy viscosity modeling relies on solving two transport equations. Those equations are used with aim to solve scalar quantities for computation of the turbulence
3.2. Curvature correction

In this thesis the so called $K - \omega$ model, has been used, mainly to provide a good suitable initial condition for subsequent LES computations. The model solves one transport equation of the turbulent kinetic energy $K$ and another for the specific dissipation rate $\omega$. Here, $\omega$ is proportional to $\epsilon/K$. A comprehensive derivation of the $K - \omega$ model can be found in the original work of Wilcox (1998). The $K$ equation is derived from the momentum equation with introduced Reynolds decomposition. Subtracting the Reynolds equation then results in an expression for the velocity fluctuation. Each component of the velocity fluctuation is multiplied by itself and then the three components are added together and the sum is time-averaged. In tensor notation the model takes the following form:

\[
\begin{align*}
\frac{\partial K}{\partial t} + \bar{u}_i \frac{\partial K}{\partial x_j} &= P - C_\mu K \omega + \nu_t \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_T \right) \frac{\partial K}{\partial x_j} \right], \\
\frac{\partial \omega}{\partial t} + \bar{u}_i \frac{\partial \omega}{\partial x_j} &= \left( C_{\epsilon 1} - 1 \right) \frac{\omega}{K} P - \left( C_{\omega 2} - 1 \right) C_\mu \omega^2 + \nu_t \frac{\partial}{\partial x_j} \left[ \left( \nu + \nu_T \right) \frac{\partial \omega}{\partial x_j} \right], \\
\text{Production} P &= -u_i'u_j' \frac{\partial u_i}{\partial x_j} = [EVM] = 2\nu_T S_{ij} S_{ij}, \\
\text{Dissipation} \omega &= \frac{\nu}{C_\mu K} \frac{\partial u_i'}{\partial x_j} \frac{\partial u_i'}{\partial x_i}, \\
\text{Turbulent viscosity} \nu_T &= \frac{K}{\omega} 
\end{align*}
\]

(3.26)

In the $K - \omega$ model the term $P$ accounts for the production of turbulent kinetic energy. The $\omega$ is a quantity that accounts for energy dissipation into heat. The last components in the $K - \omega$ equations are terms that accounts for spatial redistribution due to viscous and turbulent effects. Due to introduction of eddy viscosity modeling in the $K - \omega$ transport equations, auxiliary relations and coefficient are required for closure. These can be obtained experimentally by considering homogenous shear flows or isotropic turbulent decay scenarios, see Wilcox (1998) for more details. Reported advantages with the $K - \omega$ model is good response to pressure gradients, an imperative feature for separated flows. It offers a better wall boundary condition compared to $K - \epsilon$ models, where $K \to 0$ and $\epsilon/K \to \infty$. However $K - \omega$ is sensitive at non-turbulent interfaces at the boundary layer edge. It is known to be sensitive to free-stream conditions causing excessive production rate $P$ in shear flows. To address various issues Menter (1993) suggest several amendments to the standard formulation. He introduces a cross-diffusion term in the $\omega$-equation in an attempt to remedy the boundary layer edge problem. Moreover issues with unphysical excessive production are bounded using a dedicated limit factor. The modified formulation has been used sometimes in this thesis and is called Menter SST $K - \omega$ two equation model.

3.2. Curvature correction

There are several problems in fluid mechanics that involve turbulent flow along curved surfaces. Here one can mention applications such as diffuser flow, an essential component in centrifugal compressors. An early work with aim to characterize the effect of curvature on fully developed turbulent flow can be
credited to Wattendorf (1935). An experimental setup was developed using a curved channel of constant curvature and cross-section. Velocity profiles were measured between the inner and outer radius of the channel and the measured distributions exposed a characteristic tilted shape. In explanation, the peak of the circumferential velocity profile was seen to move closer to the inner radius of the channel. From the peak and towards the outer radius the measured data were seen to conform to a potential distribution \( u_{\theta r} = \text{constant} \). It was concluded that the flow velocity distribution were strongly influenced by curvature.

Several decades later some research studies have been targeting turbulence modeling in curved channels. Here one can mention the work by Shur et al. (2000) who propose a streamline-curvature effect in the framework of eddy-viscosity turbulence modeling. In a more recent study curvature correction with two-equation eddy viscosity models such as \( K - \epsilon \) and \( K - \omega \) has been assessed for idealized centrifugal compressor applications, c.f. Dufor et al. (2008).

In summary, curvature correction is a feature with aim to modify the turbulent energy production with respect to local flow rotation and vorticity respectively. The essential ingredient is introduction of a curvature correction factor \( f_c \) to the turbulent production term in the \( K \) transport equation. From Shur et al. (2000) the curvature correction factor is given as follows:

\[
f_c = \min\left(C_{max}, \frac{1}{C_{r1}(|\eta| - \eta) + \sqrt{1 - \min(C_{r2}, 0.99)}} \right)
\]

(3.27)

where \( \eta \) is a function of the strain-rate tensor and the rotation-rate tensor. The terms denoted with capital \( C \) and with an index are modeling constants as proposed by (Shur et al. 2000). Consequently there is a large freedom in choosing the model constants requiring extensive parameter sensitivity assessment. Moreover, small variations of the proposed default values may potentially yield significant differences in the predicted turbulent production.

The case presented by Wattendorf (1935) can be modeled as a 2D steady-state and fully developed turbulent flow in a circular channel. The outer radius is \( R_o = 0.25 \) m, the inner radius is \( R_i = 0.2 \) m, so that the channel width is \( d = R_o - R_i = 0.05 \) m. The influence from the channel depth is hence neglected since a 2D domain is considered. Since the flow is assumed periodic, a 30° degree sector of the circular channel is considered. The inlet opening of the domain is located at one end and the outlet is thus located on the other end. This means that flow is assumed axisymmetric so that the tangential velocity only depends on the radial position. For modeling purposes the openings are connected as a periodic fully developed interface with a specified mass flow rate of 1 kg/s. This corresponds to \( Re_d = 54000 \) which is within a turbulent regime. Figure 3.2a) shows the normalized circumferential velocity profile \( u_{\theta}/U_\theta \) obtained experimentally (Wattendorf 1935) and numerically using eddy viscosity modeling with and without curvature correction option.
3.2. Curvature correction

Evidently, curvature correction introduces a desired effect. It modifies the profile into a tilted distribution with the peak level shifted towards the inner channel radius, as observed in the experimental data. Without curvature correction there is no longer a tendency of an inflection point towards the inner radius and the profile is more symmetric about the center radius. As a consequence a larger difference is therefore obtained compared with the experimental observation.

One attempt to elucidate the effect of curvature correction in the framework of eddy viscosity modeling is to look at the Reynolds stress cross component distribution across the channel width. This quantity can be estimated by computing:

\[-u'v' = \nuT \frac{\partial u_\theta}{\partial r}\]  \hspace{1cm} (3.28)

In this expression \(\nuT\) is the turbulent viscosity as computed by the eddy viscosity model. Without curvature correction it is evident that the distribution is more symmetric with a zero level at the center radii of the channel, see Fig. 3.2a). With curvature correction the distribution exposes clear difference towards the inner radius. An interesting sharp gradient at approximately \(r/R = 0.825\) is manifested, which correlates with the location of the inflection point tendency as observed in the circumferential velocity component distribution in Fig. 3.2b). The reason for the sharp gradient at this location may be a consequence of the curvature correction factor expression. Since \(f_C\) evaluates a minimum function from two different terms, a smooth transition is not guaranteed.

The effect of introducing curvature correction modification to the \(K\) equation demonstrates a possibility to capture a better trend as compared to experimental data. It is evident that the curvature correction modification introduced depends on several modeling constants. A future work may naturally go further exploring
variations of the default modeling constants, as proposed by Shur et al. (2000). This may seem attractive at first sight but one quickly realizes that alteration of the modeling constants in the curvature correction framework yields many possible combinations. Thus, testing one set of modeling constants still leaves many others to explore. For this particular test case the curvature correction is seen to limit the Reynolds stress level towards the inner radius of the curved channel. This seems to correlate with a tendency for a developing inflection point. Unfortunately, this is not an observed feature in the experimental data. What is more likely to happen in a turbulent boundary layer in a curved channel is developing coherent longitudinal vortex structures, known as Görtler vortices (see e.g. Saric (1994)).

3.2.1. Boundary layer flow

It was shown that the flow in a centrifugal compressor can be subjected to large Reynolds numbers, a consequence of the low viscosity of the fluid. One may thus, as a first approximation, attempt to completely disregard from the effect of shear stresses. However, it turns out from observations that the flow do not slip on the wall. Thus, even if the viscosity is small for ambient air it cannot be neglected in a thin region close to the wall where the wall-normal gradient scales increases with the inverse of the viscosity. Depending on the size of the Reynolds number there can be vastly different flow regimes taking place in the thin so called boundary layer. If the flow is smooth and stable for disturbances it is said to be laminar. In this regime the velocity increases linearly with distance from the wall to the boundary layer edge. In a scenario with high enough flow speed disturbances are amplified and trigger a transition into a highly unstable and turbulent boundary layer flow. This particular boundary layer flow is observed to modify the velocity profile into a logarithmic shape. In reality however, the flow over e.g. the impeller blades may be subjected to a mix between laminar and turbulent states. Regardless of flow regime, the layer closest to the non-slip wall is laminar. It is identified as an extremely thin region called the laminar or viscous sublayer. Usually this layer is very difficult to capture experimentally and from a numerical point of view very costly to resolve with the grid. More insight into the stream-wise velocity variation in the boundary layer as function of the wall normal direction can be obtained from the streamwise-time averaged, momentum equation

\[
U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{dp}{dx} + \frac{\partial}{\partial y} \left( \nu \frac{\partial U}{\partial y} - u'v' \right) \tag{3.29}
\]

Equation 3.29 holds assuming 2D flow, steady-state flow, and \( Re >> 1 \). Thus, it has a limited application range and so does not completely take into account unsteady 3D flows in centrifugal compressors. Introducing a non-dimensional length \( y^+ = \frac{yu}{\nu} \), scale and velocity scale \( u^+ = \frac{u}{u_x} \) into the thin shear-layer equation gives the following relation:

\[
u^+ = \frac{1}{\kappa} \ln y^+ + B \tag{3.30}
\]
It is assumed here that the friction velocity is $u_\tau = \sqrt{\tau_W/\rho}$ and $\tau_W$ is the wall shear stress. In order to obtain Eq. 3.30 eddy viscosity modeling must be introduced for the turbulence stress term assuming that the velocity scales as $\sqrt{K}$ and that the mixing length varies linearly in the wall normal direction. This is also known as Prandtl’s mixing length hypothesis. The expression in Eq. 3.30 contains $\kappa$ - von Karman constant, and $B$ - empirical coefficient, and suggests a log-law variation. From observations this variation is not valid everywhere but typically only in a layer sufficiently far from the non-slip wall, i.e. in a region where $50 \leq y^+ \leq 200$. These are not universal bounds but rather case specific. For small $y^+ < 5$ the relationship $u^+ = y^+$ holds in the aforementioned viscous sublayer. For large $y^+ > 200$, i.e. outside the log-law region, any streamwise velocity deficit will start to mix with the bulk flow velocity. This area is also called the wake-region and found to be at wall normal distances exceeding approximately 15% of the total boundary layer thickness. In Fig. 3.3 boundary layer profiles obtained with Large Eddy Simulation at 40% and 90% impeller blade chord stations are evaluated w.r.t the law of the wall. In the figure the blended wall law is from an idea introduced by Reichardt (1951). It combines the viscous sublayer and the logarithmic region into a range entitled buffer layer. It is a concept known as all $y^+$ wall treatment. If the boundary layer flow is attached to the wall surface then the blended wall law enables a cost effective approach. The near wall region is extrapolated with a simple relation and hence allows for grid coarsening in the boundary layer. Unfortunately it may fail in recirculating and separated flows, e.g. surge operating conditions, so practically it is limited to attached flow.

Figure 3.3: Law of the wall, tangential velocity near the wall with mixing length model as compared with LES data at the impeller blade surface under steady near optimum design efficiency operating conditions
As the fluid particles move over the impeller blade imperfections due to surface roughness may perturb the flow. However, close to blade leading edges, the influence from perturbations will be limited and the initial boundary layer may be laminar. Additionally, the transition to a fully turbulent boundary layer is a slow process that may occur only over a long enough distance. For a smooth surface this would be as illustrated with the black curve in Fig. 3.3 at 40% blade chord. Towards the trailing edge, disturbances may grow substantially manifesting in a transition to a turbulent boundary layer, as illustrated with the gray 90% chord curve. However, this may only happen for sufficiently long distances. The length of the laminar part can be estimated with analogue to the hydrodynamic entry length for pipe flow, c.f. Hoffmann et al. (1996). Assuming laminar flow the entrance length \( L_{h,laminar} \) is estimated as:

\[
L_{h,laminar} \approx 0.05 Re_D D
\]  

(3.31)

where \( D \) is the pipe diameter. Assuming that \( D \approx 2 \) cm is in the same order of magnitude as the impeller blade chord and that laminar flow is only observed for \( Re_D < 2000 \). It suggest that the entire blade surface is subjected to laminar flow. In reality though due to installation effects, e.g. upstream duct piping, may introduce special turbulence characteristic levels influencing the boundary layer state. This in turn will influence the stability of the flow and also frictional losses. The size of this loss is highly dependent on the boundary layer state but more importantly the boundary layer thickness. A measure of the boundary layer thickness is the displacement thickness, defined by:

\[
\delta_1 = \int_0^\delta (1 - \frac{\rho u}{\rho_c U_e}) dy
\]  

(3.32)

If one assumes a laminar boundary layer state, then the displacement thickness corresponds to approximately 25% of the blade-to-blade pitch. In Fig. 3.3 this is \( y^+ \approx 500 \) at 40% chord and \( y^+ \approx 250 \) at 90% chord. In order to compute friction forces the shear stress must be integrated along the streamwise direction. For the laminar case the shear stress is proportional to the dynamic viscosity. Another complication is the sharper gradient near the wall, which from an experimental legacy point of view is difficult to integrate with any accuracy. For this reason the momentum loss thickness is normally preferred:

\[
\delta_2 = \int_0^\delta \frac{\rho u}{\rho_c U_e} (1 - u/U_e) dy
\]  

(3.33)

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

### 3.2.2. Boundary layer with pressure gradient

The impeller wheel boundary layer in centrifugal compressors is typically subjected to an adverse pressure gradient (APG). This corresponds to \( \frac{d p}{dx} > 0 \) in Eq. 3.29. The higher back pressure slows the boundary layer flow momentum;
a typical source for turbulence production. This mechanism was recognized by Pope (2000) who found that a positive contribution occurring from the diagonal stress terms in the kinetic energy equation is a vital reason. In very strong APG therefore, the boundary layer flow may come to a standstill (e.g. zero velocity) and even with more severe flow reversal. From an experimental point of view, this phenomenon was first documented by Ludwig Prandtl. In his pioneering contribution (Prandtl 1904) there is a sketch showing an event of boundary-layer separation. An adapted figure is provided in Fig. 3.4. A striking conclusion therefore is that there is always a potential risk for boundary layer separation over impeller blades. It is due to the growing pressure gradient toward the exducer. Moreover, there is no simple solution for preventing a separation, even in the hypothetical limit where the dynamic viscosity goes to zero. A very fundamental consequence therefore is that a significant large scale flow phenomenon is influenced by a term which is formally of second order.

Figure 3.4: Illustration showing the response of the velocity profile during a boundary layer separation. The sketch is adapted from Prandtl’s original paper.

3.3. Large Eddy Simulation (LES)

In contrast to eddy viscosity modeling, LES is a time dependent technique resolving a broad range of scales on the computational grid. Thus, it provides influence of de-alignment and rotation, which are properties of anisotropic turbulent flow especially when approaching unstable surge conditions. Eddy scales that happen to be smaller than the computational grid must be accounted for by a model. Thus, the idea is to explicitly solve a substantial part of the turbulence (expressed in terms of turbulent kinetic energy) and thereby minimizing the modeling to account for the small-unresolved flow scales. For a
3. Modeling Compressor Flow

generic flow variable, the resulting filtered variable is defined as:

\[
\bar{\phi}(t, x) = \int_V G(x - x', \Delta) \phi(x, t) \, dx'
\]  

(3.34)

where \( G(x_i, \Delta) \) is a filter width function characterized by \( \Delta = V_{cell}^{1/3} \). One difference from the Reynolds decomposition is that the filtered fluctuation variables in the LES approach are not necessary equal zero, i.e. \( \bar{\phi}' \neq 0 \). The filter is seen to rearrange the Navier-Stokes equations into a form with a similar appearance as the unsteady Reynolds Averaged Navier-Stokes equations (RANS). It is given as follows:

\[
\frac{\partial \bar{\rho} \bar{u}_i}{\partial t} + \frac{\partial \bar{\rho} \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\sigma}_{ij}}{\partial x_j} + \frac{\partial \sigma_{SGS,ij}}{\partial x_j}
\]  

(3.35)

The filtered stress-tensor represents the sub-grid scale (SGS) stresses, defined as:

\[
\sigma_{SGS,ij} = -\bar{\rho}(\bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j)
\]  

(3.36)

whereas the viscous stresses are given as:

\[
\bar{\sigma}_{ij} = -2\mu_t \bar{S}_{ij} + \delta_{ij} \frac{2}{3} \mu_t \bar{S}_{kk}
\]  

(3.37)

The SGS can be modeled, based on dimensional arguments in a form similar to the eddy-viscosity Boussinesqu closure. However, these two expressions are fundamentally different in their origin and foundation in spite of the formal resemblance. The SGS expression contains explicitly the (square) of the filter width. Thus, the SGS contribution tends to vanish as the filter size diminishes. This is not the case in the eddy viscosity model, as it is independent on the resolved scales. Thus, the SGS stresses can be expressed in analogue to the viscous stress, using the resolved strain rate:

\[
\sigma_{SGS,ij} = -\mu_t \bar{S}_{ij} + \delta_{ij} \frac{2}{3} k_{SGS}
\]  

(3.38)

where \( \mu_t \) is the SGS turbulent viscosity (often defined as \( \mu_t = \Delta^2 |\bar{S}_{ij}| \)) and \( k_{SGS} \) is the SGS kinetic energy. The precise expression for \( \mu_t \) maybe given in different ways, with the condition that it goes to zero as the square of the filter-size. As the filter size decreases, one may find that the contribution from the SGS is small enough as compared to the other terms in the momentum equation and hence may set the SGS term itself to 0 (i.e. \( \mu_t = 0 \)). It must be stressed that one cannot do the same thing with the Reynolds-stress term (i.e. neglecting it altogether).

The governing equations are cast in continuous integral form and then formulated in discrete form in finite control volumes that constitutes the computational domain. An approximate algebraic representation is obtained via Taylor series expansion. Naturally the expansion is truncated which introduces error since higher order terms are omitted. However, it has been demonstrated that spatial and temporal discretization schemes of formal second-order or higher exhibit low dissipative error, provided that the resolution is adequate.
In compressible flows, not only the dissipation error is important, but also dispersion errors (i.e. the propagation speed being wave number dependent) may be critical. Conservation is important primarily in the case of discontinuities. Otherwise the error would be of the same order as the discretization scheme. The conservation must take place only on the integral level, i.e. total mass flow should vanish to round-off and not to truncation order level, c.f. Margolin & Rider (2002) and Fureby & Grinstein (2002). It is also documented that the dissipative truncation error stresses mimic Smagorinsky-type SGS modeling for the energy equation. The mechanism of the SGS model is to dissipate kinetic energy at the smallest scales. Since this effect is obtained from the truncation error, LES with implicit SGS modeling is considered in the current work.

3.4. Advanced post-processing

A large amount of time-dependent 3D data is generated with the LES calculations of the complex flow scenario associated with the centrifugal compressor. In order to take full benefit of that, advanced post-processing methods may be necessary, in particular if one has to quantify specific instabilities arising in the fluid flow that may be crucial for understanding the compressor behavior at unstable, low mass flow rate conditions. Areas with high vorticity, defined as the curl of the velocity $\nabla \times u_i$ can be used to identify regions with coherent motion, e.g. large scale vortical structures associated with rotating stall cells. The vorticity transport equation, derived by taking the curl of the momentum equation, can be considered:

$$\frac{\partial \omega}{\partial t} + (u_i \cdot \nabla)\omega = (\omega \cdot \nabla)u_j - \omega \nabla \cdot u_j + \frac{1}{\rho^2} (\nabla \rho \times \nabla p) + \nu \nabla^2 \omega \quad (3.39)$$

The first term on the right hand side is a mechanism for stretching or tilting of vorticity because of velocity gradients. The second term on the right hand side represents vorticity generation due to compressibility effects, which can be shown by using the continuity equation. The third part on the right hand side is referred as a baroclinic term. It is associated with vorticity production due to non-aligned density and pressure gradients. The last term is due to viscous dissipation. In high $Re$ number flow it has a negligible effect on structures that lack large gradients.

If the compressor is installed with an upstream bell mouth inlet, then streamlines of the flow would be seen to be funneled towards the impeller with a slight convergence directed towards the center line. The axial velocity increases due to flow acceleration. This causes a reduced pressure just upstream of the impeller. It can be seen as a flow expanding with $\nabla \cdot u_j > 0$. Vorticity is governed by Kelvin’s theorem, which dictates that $|\mathcal{O}|$ grows when a vorticity tube cross section reduces. In the diffuser on the other hand, increased cross sectional area reduces the velocity and the pressure goes up. This is a flow compression, i.e. $\nabla \cdot u_j < 0$.

The energy transfer in the turbulent cascaded is by large governed by the vortex stretching term $(\omega \cdot \nabla)u_j$. For example, the longitudinal distance between
two points on a vortex will tend to grow over time. In a statistical average sense this cause increased vorticity in the elongated direction and reduced vorticity in the orthogonal directions. However, the angular momentum of a stretched vortex is conserved, and proportional to \( \omega r^2 \), where \( r \) is the vortex radius. The kinetic energy associated with the vortical motion scales as \( \omega^2 r^2 \). If the angular momentum is conserved there must be a reduced kinetic energy, i.e. an energy transferred from large to small vortical structures.

Another option for extracting coherent vortical features of the flow is via the \( \lambda_2 \) criteria. It can potentially capture structures associated with tip clearance vortices, or horse shoe vortices at the blade/hub and even structures caused by boundary layer separation. The \( \lambda_2 \) criteria is computed from the second eigenvalue of \( (S^2_{ij} + \Omega^2_{ij}) \), where \( S_{ij} \) is the strain-rate tensor and \( \Omega_{ij} \) is the spin tensor. One problem with assessment of the instantaneous flow field using either the vorticity or the \( \lambda_2 \) criteria is the inherent complexity of the flow. Therefore, the large flow structures must be filter out from the chaotic flow field to enable any interpretation.

### 3.4.1. Fast Fourier Transformations (FFT)

The frequency content of a signal can be obtained by computing the power spectra density flow quantities of interest in one or preferably several points. This is obtained by using a time Fourier transform for a real valued signal \( h(x, t) \):

\[
FT_t[h(x, t)] = \int_{-\infty}^{\infty} h(x, t)e^{-i\omega t} dt = g(x, \omega), \forall \omega \in [-\infty, \infty] \tag{3.40}
\]

here the real valued function may be a pressure or a velocity component, which depends on space \( x \) and time \( t \), and where \( \omega \) is the angular frequency. This yield a frequency-domain description of the signal and the power density function is obtained by:

\[
PSD(\omega) = \lim_{T \to \infty} E[|g_T(x, \omega)|^2] \tag{3.41}
\]

where \( E \) denotes the time-averaged value, and subscript \( T \) means integration is over a finite period in practice. For a centrifugal compressor some specific frequencies will be subjected to large PSD values, for instance the blade passing frequency. Once interesting narrowband frequencies have been identified it is just a matter of repeating the procedure for all points in the computational domain. However, this requires sufficient random access memory on the computer so in practice the number of data points are restricted by considering only some few post-processing planes. Another option is to utilize the time-space Fourier transform. It may be considered as one transform in time and then another in space in a sequential fashion. It is given by:

\[
FT_{ts}[h(x, t)] = \int_S \left( \int_{-\infty}^{\infty} h(x, t)e^{-i(\omega t + kx)} dt \right) dx = f(k, \omega), \forall k \in K, \omega \in [-\infty, \infty] \tag{3.42}
\]
3.4. Advanced post-processing

This transform is useful because multiplying it with its complex conjugate gives the frequency-wavenumber autospectrum defined as:

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

(3.43)

Commonly this technique is adopted for assessing the group velocity of propagating wave disturbances along some line of interest. It should be realized that multiple waves with different frequencies and wavelengths may interfere and develop an envelope (or modulation in space). It is well known that the phase velocity is given as the wavelength times frequency

\[ v_\theta = f\lambda \]  

(3.44)

The wave number is defined as

\[ k = \frac{2\pi}{\lambda} \]  

(3.45)

Therefore,

\[ v_\theta = \frac{2\pi f}{k} \]  

(3.46)

If the frequency-wavenumber autospectrum reveals any sharp lines with a distinct slope it corresponds to the group speed of the propagating disturbances along the line

\[ v_g = 2\pi \frac{\partial f}{\partial k} \]  

(3.47)

The Sound Pressure Level (SPL) acoustic spectra are extensively used to indicate the local change in pressure from the reference ambient pressure that a sound wave causes. It is computed as follows:

\[ SPL = 10 \log_{10} \left( \frac{p_{rms}^2}{p_{ref}^2} \right) \]  

(3.48)

where \( p_{ref} \) is the reference sound pressure and \( p_{rms} \) is the root mean square sound pressure.

3.4.2. Two-point cross correlation

The two-point cross correlation is an alternative to the frequency-wavenumber autospectrum. It enables computation of the group speed of propagating disturbances along a line but with a time-domain description. Along a specific coordinate (say \( x \)) it is given by:

\[ R(x, \tau) = \frac{< u(x_{ref}, t_{ref})u(x, t_{ref} + \tau) >}{\sqrt{< u(x_{ref}, t_{ref})^2 >^2} \sqrt{< u(x, t_{ref} + \tau) >^2}} \]  

(3.49)

If evaluated for all points on the line, some of them may show strong correlation. A strong correlation is expected when two signals overlap. Another possibility is if the phase shift of two signals corresponds to something close to the wavelength or the wave period. In case of a flow acoustic scenario, in principle a clear pattern with inclined lines may appear. The slope would then correspond to the local convection speed of the propagating disturbances.
3.4.3. **Proper Orthogonal Decomposition and Dynamic Mode Decomposition**

A common representation of the flow field variables is that they can be represented as a finite sum of temporal $a(t)$ and spatial $\phi(x)$ mode contributions.

$$u(x) \approx \sum_{j=0}^{J} a_j(t)\phi_j(x)$$  \hspace{1cm} (3.50)

One merit is that the temporal and spatial parts may be analyzed separately. If the modes are sorted with respect to their energetic content, it may enable assessment of leading order modes and their level of significance. Thus, if all modes of low significance are truncated it would allow investigating the flow, without the complexity of small scale turbulence contaminating the large scale coherent flow structures.

Proper Orthogonal Decomposition (POD) is a post-processing methodology adopting this idea where coherent structures are extracted based on correlation of considered flow field quantities, c.f. Lumley (1970). One assumption is that the modes must satisfy spatial orthogonality. POD decompose the flow field into temporal $a_j(t)$ and spatial $\phi_j(x)$ components:

$$u(x,t) \approx a_0(t)\phi_0(x) + \sum_{j=1}^{\infty} a_j(t)\phi_j(x),$$  \hspace{1cm} (3.51)

where $a_j(t)$ contains the temporal and $\phi_j(x)$ contains the spatial information, respectively. Here the time-averaged flow corresponds to the zeroth indexed mode. The aim with POD is to find modes in such a way that spatial modes are orthogonal and that the flow field in a least square sense is optimally represented:

$$\min_{\phi_j} \left( u(x,t) - \sum_{j=0}^{J} a_j(t)\phi_j(x) \right)^2$$  \hspace{1cm} (3.52)

For determination of the POD modes, an ensemble of the flow field data is constructed. It consists of a snapshot matrix with $M$ spatial grid points and $N$ times given as:

$$U = [u_1, u_2, ..., u_N]$$  \hspace{1cm} (3.53)

where the snapshots are sampled at equidistant time-steps. Since the matrix $U$ can be very large for LES applications a computational efficient procedure is using the Singular Value Decomposition (SVD), which gives:

$$U = V\Sigma W^T$$  \hspace{1cm} (3.54)

This produces a diagonal matrix $\Sigma$ with same dimension as $U$, and with non-negative elements in decreasing order. The unitary matrices $V$ and $W$ are orthonormal and contains the eigenvectors. Thus, the spatial modes $\phi_j$ are determined by the columns of $V = UW\Sigma^{-1}$, and the temporal mode coefficients $a_j$ are given by the rows in $\Sigma V^T$. One limitation with the method is that a particular mode shape cannot be specifically linked to a particular frequency.
Dynamic Mode Decomposition (DMD) is another flow decomposition option. For an introduction, the reader is directed towards the work by Schmid (2010). With DMD some of the shortcomings with the previous method are amended by including the dynamic behavior so that modes can be associated with a particular frequency. Therefore it is a suitable choice to flow applications subjected to tonal behavior, see Alenius (2014). For DMD it is assumed that the flow field can be represented by spatial modes $\phi_j$ and more that it can be represented in a linear sequence of discreted time instances. In an iterative mapping for future time where $m$ starts at time zero the flow field is given as:

$$u(x, m\Delta t) = \sum_{j=1}^{\infty} \eta_j \lambda_j^m \phi_j(x) = \sum_{j=1}^{\infty} \eta_j e^{(\beta_j - i\omega_j)m\Delta t} \phi_j(x)$$

(3.55)

where $\beta_j$ - the growth rate, $\omega_j$ - is the frequency, and $\eta_j$ - the mode amplitude. Here, $\lambda_j$ is the eigenvalue of the linear operator, which represents the temporal evolution of the dynamical system. The effect of the linear operator can here been seen to operate on a sequence of state vectors. The state vectors are just the assembly of observed time-dependent snapshots of the flow field, i.e. similar to the POD method. The operator is assumed to be linear and it is also assumed that the dynamic flow process is contained in the temporal information. The frequency of the mode shape is determined by:

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

(3.56)

The growth rate of the mode shape is computed as:

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

(3.57)

A reconstruction in time may subsequently be obtained using:

$$u_j(x, t) = \text{Re} \left( \eta_j e^{(\beta_j - i\omega_j)t} \phi_j(x) \right)$$

(3.58)

A problem can occur if the sampling frequency $1/\Delta t$ is not chosen appropriately for the underlying dynamic flow process. If violated there may be issues with aliasing. According to the sampling theorem, if there are at least two samples per period, then aliasing may be avoided. However, Schmid (2010) recommends using a sampling frequency that is a factor six times larger as compared to the highest frequency of interest.

### 3.5. Ffowcs Williams and Hawkings equation

The Ffowcs Williams and Hawkings (FWH) equation is one option to reduce the computational cost associated with directly resolving the acoustic sound field due to moving sources, see Ffowcs Williams & Hawkings (1969). In brief, the FWH equation is a reformulation of Lighthill’s acoustic analogy, but made applicable also for moving bodies. Thus, it enables assessment of a wider range of applications, e.g. propeller noise problems (Brentner & Farassat 2003; Farassat 2001). The major moving component in a centrifugal compressor is
the impeller, which as a first approximation can be modeled as a rigid body motion. The impeller blade surface is described by a function

\[ f(\mathbf{x}, t) = \begin{cases} > 0, & \mathbf{x} \notin V \\ = 0, & \mathbf{x} \in S \\ < 0, & \mathbf{x} \in V \end{cases} \]

(3.59)

It is zero on the moving surface \( S \), negative within the volume \( V \) of the rigid body, otherwise it is positive. Using the Heaviside function with the body surface function \( f \), then it can be shown that the mass and momentum equations takes the following form:

\[ H(f) \left[ \frac{\partial \rho'}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_j) \right] = 0 \] (3.60)

\[ H(f) \left[ \frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j + p' \delta_{ij}) \right] = 0 \] (3.61)

where \( \rho' \) and \( p' \) are assumed small fluctuation quantities. In this formulation it is seen that viscous stress is neglected and so are all external source terms. If one considers a point on the moving surface the total rate of change of its Heaviside function must be zero:

\[ \frac{D_s H(f)}{Dt} = 0 \Rightarrow \frac{\partial H}{\partial t} + \mathbf{V} \cdot \nabla H = 0 \] (3.62)

Here, \( \mathbf{V}(\mathbf{x}_s, t) \) is the body surface velocity. Inserting this knowledge into equations 3.60 and 3.61, the following is obtained:

\[ \frac{\partial}{\partial t} (\rho' \mathbf{H}) + \frac{\partial}{\partial x_j} (\rho u_j \mathbf{H}) = \rho_0 V_j \frac{\partial \mathbf{H}}{\partial x_j} + \rho (u_j - V_j) \frac{\partial \mathbf{H}}{\partial x_j} \] (3.63)

\[ \frac{\partial}{\partial t} (\rho u_i \mathbf{H}) + \frac{\partial}{\partial x_j} (\rho u_i u_j \mathbf{H} + p' \delta_{ij} \mathbf{H}) = \rho u_i (u_j - V_j) \frac{\partial \mathbf{H}}{\partial x_j} + p' \frac{\partial \mathbf{H}}{\partial x_i} \] (3.64)

These last two equations can be combined using \( \frac{\partial(3.63)}{\partial t} - \frac{\partial(3.64)}{\partial x_i} \) giving the FWH equation in a general form:

\[ \frac{\partial^2 (\rho' \mathbf{H})}{\partial t^2} - \nabla^2 (\rho' \mathbf{H}) = \frac{\partial^2 (\rho u_i u_j \mathbf{H})}{\partial x_i \partial x_j} + \frac{\partial}{\partial x_i} \left[ \left( \rho_0 V_j + \rho (u_j - V_j) \right) \frac{\partial \mathbf{H}}{\partial x_j} \right] \]

(3.65)

A simplified form resembling a wave equation is obtained by assuming adiabatic changes of state and by using the relation \( c_0^2 = p' / \rho' \). Moreover if rigid body rather than arbitrary motion is assumed, i.e. \( \mathbf{u} \cdot \mathbf{n} = \mathbf{V} \cdot \mathbf{n} \), then the reduced FWH equation is given as follows:

\[ \left( \frac{1}{c_0^2} \frac{\partial^2}{\partial x_i^2} - \nabla^2 \right) p' \mathbf{H} = \frac{\partial}{\partial t} \left( \rho_0 V_i n_i |\nabla f| \delta(f) \right) - \frac{\partial}{\partial x_i} (p' n_i |\nabla f| \delta(f)) + \frac{\partial^2}{\partial x_i \partial x_j} (\rho u_i u_j H(f)) \]

(3.66)

The left hand side is the familiar wave equation for a small pressure disturbance \( p' \). On the right hand side there are three different source terms. The first source term on the right is the thickness ("monopole") noise due to unsteady volume displacement of the fluid due to the acceleration of the solid surface.
Since most impellers are designed with thin blades, i.e. their volumes are small, it means that the thickness noise is often negligible for centrifugal compressors. The second source term on the right hand side of Eq. 3.66 is the blade loading ("dipole") acoustic source term. It is associated with unsteady pressure fluctuation on the moving surface. This term is typically the major noise source for centrifugal compressors. The last source term in Eq. 3.66 is related to unsteady Reynolds stress or transport of momentum. It is also referred to as a quadrupole source, but usually insignificant in comparison with the blade loading term for centrifugal compressor applications. The goal is then to integrate Eq. 3.66 under assumption of no obstacles between the source and the receiver location. For a more detailed discussion see the works by Farassat (1981).
4.1. Discretization

The governing equations (3.1, 3.2 and 3.3) together with the constitutive relations describe an inherently complex system consisting of non-linear partial differential equations. A major difficulty is the lack of analytical solution(s) for this non-linear system of equations. One may find particular solutions to the system by making significant assumptions and simplifications. If such simplifications and assumptions yield too poor description of the flow, one have no other option than using numerical methods to solve the equations, which are believed to describe correctly the physics of the flow. Numerical solutions require discretizing the domain (space-time) under consideration along with the equations. These discretization steps may be done in different ways: the volume cell used in the computational grid may for example have different shapes with different number of faces. Similarly, the discretization of the equations can be done using different approximations. A common approach is to use local polynomials to approximate the behavior of the dependent variables. The discrete approximations can be derived, using various formulations. The bottom line is that the different approaches yield an approximation to the equations with a residue that is proportional to a power of the characteristic size of the grid. For example finite differences and finite volumes belong to this class of discretizations. An introduction to numerical methods may be found, for example in Ferziger & Peric (2012). In the discretization conservation laws, it is natural to go back from the differential formulation to the conservation formulation for a control volume that has the same form as a computational cell. A variable \( \phi \), representing specific mass, momentum and energy is introduced. The conservation laws in vector notation are:

\[
\frac{\partial}{\partial t} \int_V \rho \phi dV + \int_A \rho \phi (v - v_{sp}) \cdot da = \int_A \Gamma_\phi \nabla \phi \cdot da + \int_V S_\phi dV
\]

(4.1)

where for mass balance \( \phi = 1 \), and for momentum balance \( \phi = u, v, w \), and for energy balance \( \phi = e + \frac{1}{2}v^2 \). Thus, there is integration over individual finite volume cells, where \( a \) is the surface vector of the control volume \( V \). The terms \( \Gamma_\phi \) and \( S_\phi \) are associated diffusion and source coefficients, which can be deduced from the parent equations. One merit of this formulation is that
the discretized forms preserve the conservation characteristic of the parent differential equations. In case of rigid body motion a grid velocity term $v_{sp}$ is included for the convection, i.e. the 2nd term on the left hand side. This velocity is simply the relative velocity between the fluid ($v$) and the surface ($v_{sp}$). The first term in Eq. 4.1 is the transient term, which is included in LES but is omitted in steady-state RANS computations. The second term expresses the flux of each variable through the face area $a$ of the control volume $V$ due to the flow and due to the motion of the face of the control volume, respectively. The first term on the right hand side is associated with transport of each variable due to diffusion. For steady state simulations the grid is stationary with $v_{sp} = 0$. In order to model a rotation due to the impeller motion, one may incorporate a volume (Coriolis) source in the momentum equation.

$$S_\phi = \rho \omega \times v_{rel} \quad (4.2)$$

The balance equation (Eq. 4.1) is cast into a discrete form using a cell-centered control volume formalism. In discrete form for a cell index $O$ it is given by:

$$\frac{\partial}{\partial t} (\rho \phi V)_0 + \sum_f [\rho \phi (v - v_{sp}) \cdot a]_f = \sum_f (\Gamma_\phi \nabla \phi \cdot a)_f + (S_\phi V)_0 \quad (4.3)$$

where the summation is over all faces $f$ of the control volume. Each item in the discrete form of the transport equation is subsequently approximated with a discretization scheme. Throughout the thesis, the transient term is approximated with an implicit 2nd order scheme given by:

$$\frac{\partial}{\partial t} (\rho \phi V)_0 = \frac{3[(\rho \phi V)_0^{n+1} - (\rho \phi V)_0^n] + [(\rho \phi V)_0^n - (\rho \phi V)_0^0]}{6\Delta t} \quad (4.4)$$

The next term of interest is the convective term. A higher order scheme will give more accurate result. However, higher order schemes may be sensitive and unstable. A good compromise between accuracy and robustness is the 2nd order upwind/central scheme, given as:

$$(\hat{m}_f \phi)_f = \begin{cases} \hat{m}_{f,0} \phi_{f,0} + (1 - \sigma)[f \phi_{f,0} + (1 - f)\phi_{f,1}] & \text{for } \hat{m}_f \geq 0 \\ \hat{m}_{f,1} \phi_{f,1} + (1 - \sigma)[f \phi_{f,0} + (1 - f)\phi_{f,1}] & \text{for } \hat{m}_f < 0 \end{cases} \quad (4.5)$$

where $\sigma$ is a blending factor between upwind or central. Subscripts 0 and 1, respectively, denote two neighboring cells. The face values are then linearly interpolated from cell values on either side of the face using reconstruction gradients, via:

$$\phi_{f,0} = \phi_0 + (x_f - x_0) \cdot (\Delta \phi)_0$$
$$\phi_{f,1} = \phi_0 + (x_f - x_0) \cdot (\Delta \phi)_1 \quad (4.6)$$

If the solution is well behaved then it is possible to use higher order schemes, such as the Hybrid MUSCL 3rd-Order/Central-Differencing scheme. One alteration to the 2nd order scheme is usage of a larger stencil but similar in the sense that the $\sigma$ switch controls the amount of blending with MUSCL 3rd-order upwind compared to 3rd-order central. Detailed introductions of these schemes can be found in Darwish & Moukalled (1994) and Demirdzic et al. (1993).
Finally, the gradient of fluid property $\phi$ in the diffusive flux term is obtained with a 2nd order scheme

$$\nabla \phi_f = (\phi_1 - \phi_0) \frac{a}{a \cdot (x_1 - x_0)} + \nabla \phi - (\nabla \phi \cdot (x_1 - x_0)) \frac{a}{a \cdot (x_1 - x_0)} \quad (4.7)$$

where $\nabla \phi = (\nabla \phi_1 + \nabla \phi_0) / 2$. It can be seen that the discretization employed yields an algebraic system of the non-linear equation system, due to the coupling of the dependent variables within the system. The algebraic system can be linearized by “freezing” the coefficients in between the iterations. For the transported variable $\phi$ at iteration $k + 1$ becomes:

$$a_p \phi^{k+1}_p + \sum_n a_n \phi^{k+1}_n = b \quad (4.8)$$

where summation is assumed over all the neighbors $n$ of cell index $p$. The right hand side, denoted $b$ is the explicit result obtained at iteration $k$. The coefficients $a_p$ and $a_n$, respectively, are a result from the discretization process. From this point the transported variable at the next time iteration can be obtained by using Gaussian elimination or LU decomposition. However, for centrifugal compressor applications, the system is very large making matrix inversion procedures impractical. Instead the unknowns $\phi^{k+1}$ are cast into a correction ($\Delta$) form given as:

$$\frac{a_p}{\omega} \Delta \phi_p + \sum_n a_n \Delta \phi_n = r \quad (4.9)$$

where $\Delta \phi$ is the difference in the transported variable at current $k + 1$ and previous $k$ iteration, respectively. The term on the right hand side $r$ is called residual, given by:

$$r = - \left[ \frac{\partial}{\partial t} (\rho \phi V) + \sum_f (\rho \phi v \cdot a_f - \sum_f (\Gamma \nabla \phi \cdot a_f - S_{\phi} V) \right] = 0 \quad (4.10)$$

The residual represents the derivation of the discretized form of the original equation at iteration $k$ from 0 (i.e. the error in solving the discrete equation). When the discretized equation is satisfied the residual goes to zero. In the thesis, the Gauss-Seidel implicit iterative solution procedure is used. This method updates the transported value in each cell using information from cell neighbors. The Gauss-Seidel can be shown to have a faster convergence rate as compared to the Jacobi method, which is why it is preferred. However, Gauss-Seidel alone is subjected to a very poor computational turnaround. For a discrete problem with $N$ degrees of freedom (roughly equal the number of grid points) the total work for convergence scales as $N^2 \ln N$. Thus, for increasing problem size the computational effort increases more than quadratically. The idea is that Gauss-Seidel enables efficient smoothing of local errors. However, it suffers on convergence because it takes time for boundary information to propagate into the domain. Instead of a large number of Gauss-Seidel sweeps, we use an Algebraic Multigrid (AMG) algorithm. The concept of the Multigrid algorithm is to handle the error in the solution on a sequence of coarser grid levels.
4.1. Discretization

Through this process the boundary information propagates much faster on a coarse grid as compared to a fine grid and thereby the solution is attained much faster. The AMG mechanism is thus to alternate the Gauss-Seidel iteration between coarse and fine grid levels. This is accomplished by means of transfer operators between coarser and finer grids. Optimally, AMG converges with a computational work proportional to $N \ln N$, which implies linear growth with increasing problem size.

The stability condition criterion stipulates that the region of dependence of the discretization computational volume should include the region of the dependence of the partial differential equations. This translates into the condition that information propagating on the grid should not move more than one cell length at a time per time-step. It is a necessary condition and governed by the local convective CFL number (Courant-Friedrich-Lewy), which for compressible flows can be defined:

$$CFL = \frac{|(U + c)\Delta t|}{\Delta x} \leq 1 \quad (4.11)$$

It is a function of a characteristic velocity $U$, the time-step size $\Delta t$, the characteristic cell with size $\Delta x$, and where $c$ is the sound speed. The highest speeds for rotating machinery problems are typically found near the blade tip region. A subsequent conservative time-step estimate for stability is given as:

$$\Delta t = \frac{(\Delta x)_{tip}}{\omega R_{tip}} \quad (4.12)$$

where $\omega$ is the impeller angular velocity.

4.1.1. Boundary conditions

Since a centrifugal compressor generates noise, special considerations are needed to allow pressure fluctuation disturbances to propagate freely at the sound speed towards the farfield. If the centrifugal compressor is fitted with a bell mouth inlet, the upstream flow may be approximately quiescent where stagnation temperature and pressure are given from ambient reference conditions. To allow sound propagation then, the correct treatment is to adopt a non-reflective boundary condition. A detailed description of non-reflective boundary treatments can be found in the works of Giles (1990) and Saxer (1992). In short, the freestream velocity on the boundary is corrected by a combination of the freestream velocity and the boundary normal velocity component:

$$\vec{v}_f = \vec{v}_\infty + (v_{fn} - v_\infty) \frac{\vec{a}}{|\vec{a}|} \quad (4.13)$$

where $v_\infty$ is the boundary normal freestream component. From characteristics the boundary normal velocity $v_{fn}$ is obtained:

$$v_{fn} = \frac{1}{2}(v_{0n}^r + v_\infty) + \frac{c_0 - c_\infty}{\gamma - 1} \quad (4.14)$$

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

\[
c_0 = \sqrt{\gamma RT_0}
\]  

(4.15)

In case of subsonic inflow, the freestream pressure via the isentropic relation is utilized for computing the boundary pressure:

\[
p_f = p_0^r \left( \frac{T_f}{T_0} \right)^{c_p/R}
\]  

(4.16)

where the temperature on the boundary face is given by:

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

(4.17)

and where:

\[
c_f = \frac{1}{2}(c_\infty + c_0) + \frac{1}{4}(\gamma - 1)(v_0^r - v_\infty n)
\]  

(4.18)

It should be noted that errors can occur with this treatment in the non-normal direction, i.e. where the boundary is not normal to the incident compression waves.

In the experimental setup (Guillou et al. 2012) of the centrifugal compressor investigated in this thesis the outlet mass flow is regulated with a lock valve. Such device can influence the flow in such a way that incoming pressure waves are being reflected. One option to include this effect is to consider a mass flow rate boundary condition. The boundary inputs are a total mass flow rate, total temperature and flow direction (in case of flow reversal). As the compressor operation is towards its surge line, flow reversal may occur. This may potentially yield some side effects in scenarios with mixed inflow and outflow on the outlet boundary surface. From a numerical stability point of view provisions should be taken to ensure 100% outflow in the boundary normal direction, and this is guaranteed with the mass flow rate boundary condition type.

The specified total mass flow rate is distributed over all faces of the boundary and a uniform mass flow rate is calculated on each face of the boundary:

\[
\dot{m}_f = \dot{m}_{total} \frac{|a_f|}{\Sigma_f |a_f|}
\]  

(4.19)

The static pressure and static temperature, respectively, at the boundary are extrapolated from the interior of the domain. It can be mentioned also that the current compressible flow solver has been used previously in the research group Competence Center for Gas Exchange (CCGEx) for solving several fluid flow problems associated with gas-exchange processes and turbocharging in internal combustion engine applications, see e.g. Semlitsch et al. (2014b, 2015); Semlitsch & Mihăescu (2016).
Chapter 5

Results: Highlights and Discussion

An overview of the centrifugal compressor theory and expected fluid instability phenomena occurring under off-design conditions have been discussed briefly in the previous chapters. Part II of the thesis contains selected papers showing the main results. Highlights from those papers are outlined in the following together with listing of main researching findings.

In this thesis the main application under consideration has been a vaneless diffuser centrifugal compressor fitted with a ported shroud, as shown in Fig. 5.1 and presented in Papers 1 to 5. However, the same computational tool has also been applied for investigating the stall instability emerging in a vaneless diffuser of a larger centrifugal compressor, without explicitly considering the impeller presence in the simulations. Instead, the effect of the impeller blade passing is provided as meridional and tangential velocity profiles, respectively. This case is presented in Paper 6.

In order to fully characterize the compressor behavior and the developed flow instabilities at off-design conditions, there is a need to accumulate data for a large number of operating conditions. However, one have to keep in mind that if the target is only to characterize the overall compressor performance (pressure ratio and efficiency) as a function of mass flow rate, steady-state RANS based simulation is the method of choice (see e.g. Sundström et al. (2017a)). If the purpose, as previously stated, is to look into understanding the flow instabilities developed at off-design/unstable operating conditions, and their impact on the compressor’s performance and noise generation, the LES calculation is the method of choice. Naturally, due to the required large computational resources with such an approach, it is not feasible to cover the entire compressor map using LES. For this reason, only selective operating conditions are considered under stable and unstable regimes. This choice was made for two different speed-lines, c.f. Papers 1 and 3 for further details.

As with any numerical simulation it is always relevant to check the validity of the computed result. Accuracy typically depends on modeling assumptions but equally important is the influence from the employed discretization. Thus, one may distinguish between errors and uncertainties in solving the discrete equations, errors in solving the model (PDE) equations and finally the inherent uncertainties and possible errors in the model equations themselves. For
5. Results: Highlights and Discussion

Figure 5.1: Side and front views of the centrifugal compressor are shown at the top of the figure. Location of planes used for data post-processing and locations of the pressure sensors D0 and IS1 are indicated. The IS1 point is located in the middle of the ported shroud cavity 0.22\(D_2\) from the diffuser back wall and oriented 127\(^o\) clockwise from the vertical. The D0 point is located mid of the diffuser channel at radii 55.5 mm and oriented 30\(^o\) clockwise from the vertical. The impeller section intersects the impeller region at 50% blade span. The P1 and P2 planes are located 0.85\(D_2\) and 0.76\(D_2\), respectively, from the back wall, and oriented with perpendicular intersection of the inlet duct. Subfigures in the bottom part of the figure depict inlet and outlet boundaries of the computational domain. The inset detail at the bottom right illustrates the blade passage grid resolution at 50% blade span.

historical reasons the scientific community have suggested stages to assess the relevance of numerical results, c.f. e.g. Roache (1998). One necessary step is termed verification. This is a kind of certification process where implemented routines in the CFD code produce expected result. For example, if the governing Navier-Stokes equations are drastically simplified, e.g. steady-state, 1D/2D, incompressible flow, for an idealized problem scenario, then there can be an
exact analytical solution available. The code is therefore setup for such flow scenario and with help of the exact analytic solution the true error is computed. Of course if an analytic solution exists then there is very little need to employ CFD. However, for complex problems, e.g. the unsteady flow in a centrifugal compressor at low-mass flow rates, such an analytical flow solution does not exists. A possibility is to complete a verification of the calculation by assessing the flow solution’s sensitivity with respect to the numerical, grid and boundary setups employed, respectively. The other option is called validation, which relies on surveying the relative difference compared to some experimental measurement. One problem with validation is that the measured result depends on many parameters and is subjected also to errors. Moreover, most boundary conditions on the experimental setup are unknown. For these reasons, reasonable good agreements between experiment and numerical simulations for the considered scenarios in the current work is challenging.

Any numerical approach relying on the finite volume method is grid dependent. Therefore, one idea for checking relevance is to make gradual grid refinements, i.e. temporally and spatially, and check if relevant quantities approach an asymptotic value. In theory, a fine grid is less diffusive and would yield more accurate approximations to the solution of the governing equations as compared to a coarse grid. Based on this, the strategy in Paper 1 is to compare the result from three different grids, with different grid refinement levels. With consistent discretization schemes the truncation error should tend to reduce monotonically as the grid size is refined. However, from practical considerations, there is another issue known as round-off error due to finite precision of the computer, which may set a lower limit on the grid resolution. In practice, double precision (64-bit) is adequate for practically all CFD computations.

The main outcome from the validation process is the demonstration of the capability with LES to consistently predict the global performance parameters for a wide range of operating conditions, as shown in Fig. 5.2. Overall, the results of the numerical setup are reasonably close to the corresponding experimental data. In the figure the vertical and horizontal bars indicate variations in the monitored signals. Therefore, they are almost not visible for Case D, since this is approximately a steady-state flow condition. However, as conditions approach low mass flow rates, a substantial increase can be seen. For Case A, i.e. the most restricted mass flow rate, a limit surge cycle is manifested, associated with significant vibrations of the performance parameters.

Regarding the direct comparison with the experimental data; there are several sources causing differences. First, measurements at surge operating conditions are challenging due to the large oscillations in the system. Note that the numerical and experimental set-ups are not perfectly identical, being impossible to provide complete boundary conditions to the CFD from experiments. Moreover, one has to keep in mind that the experimental flow regulation system and the driving turbine have not been modeled in the simulations. These are
some of the reasons for which a perfect match cannot be obtained between CFD and experiments.

Naturally, there are uncertainties with numerical accuracy. For instance, discretization errors emanate from approximations to the governing equations, e.g. implicit 2nd order temporal and implicit 3rd order bounded central differencing. The dominant error terms of the finite volume method can be related to dominant error terms also with a finite difference equation, which is easier to assess analytically. For example, a finite difference equation may be assessed by substituting the Taylor series expansion back into the finite difference equation. After rearrangement the resulting so called modified equation is obtained, c.f. Hoffmann et al. (1996). By considering a simple problem e.g. the linear wave equation in one-space dimension various aspects of the finite difference formulation may be investigated, such as order of accuracy and stability requirements. If the Crank-Nicolson implicit scheme (2nd order in both space and time) is substituted into the 1D linear wave equation the modified equation contains a leading odd-order derivative term, which is being truncated. This can give rise to dispersion errors. A possible side effect from dispersion errors can be introduction of phase errors and related issues with exactly matching the measured surge frequency tonality. If however, an alternative 2nd order scheme such as the explicit Lax is considered, the formulation may yield a leading even-order derivative term. Such truncation terms are associated with dissipation errors and can contribute to amplitude errors, i.e. the initial wave height is diffused over time.
One of the best ways to check the relevance of the numerical approach is to assess if the numerical grid resolution employed is sufficient for capturing essential features of the flow. For this purpose three grids named Coarse, Medium and Fine are run for a reference operating condition. Here selected for \( \omega = 64000 \text{ rpm} \) at near maximum efficiency condition \( \dot{m} = 0.28 \text{ kg/s} \). To see a clear trend due to refinement, a factor of at least two should be employed, see Tab. 5.1. A Courant number unity is not required everywhere in the domain when implicit time stepping is employed. Ideally, for accurate time dependent solution the CFL condition should be obeyed. However, if the grid resolution is well scaled to the problem then the time-step should be such that the local CFL number should be much smaller than unity to allow convergence of the Taylor series (of the discretization) and to have the error being composed mainly of the leading neglected term. In any event, the CFL condition is relevant to ensure that information propagates correctly, for example across the sliding interface between rotating and stationary regions in the computational domain. For a fair comparison the time-step size is adjusted so that the local Courant number is similar between grids.

<table>
<thead>
<tr>
<th>grid</th>
<th>cell count</th>
<th>( \Delta \bar{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 5.1: Characteristics of the Coarse, Medium and Fine grids is listed. The second and third columns tabulates the cell count and the average cell edge length \( \Delta \bar{x}_i \) for each grid. The chosen time-step \( \Delta t \) used is presented in the last column. It is given as the number of degrees of impeller rotation per time-step.

It should be noted that a grid sensitivity procedure could be done in many different ways. One option is to extrapolate a solution to an infinitely small grid size by means of Richardson’s extrapolation. Richardson’s technique can be used to find out existence of an asymptotic behavior of the solution. This may enable increasing the accuracy (and formal order) without the need to evaluate the solution on an even finer grid. For best outcome it is suggest to state the quality in terms of several different parameters, see e.g. Celik et al. (2008). For practical reasons the number of analyzed quantities are limited. Here, one preferably chose quantities where experimental data is available. In Paper 1 experimental data was available with respect to the static pressure distribution at two different radii, see Fig. 5.3.

The essential outcome here is that the Coarse grid depicts some variations, especially at 3 o’clock where the trend with the experiential data is different. As the spatial resolution is improved the trend w.r.t. the experimental data improves. The differences between grid levels can be utilized to assess a relative
Figure 5.3: Static pressure distribution in the diffuser obtained experimentally (Guillou et al. 2012) and numerically (LES) with grids Coarse, Medium and Fine. The operating conditions corresponds to Case D at 64000 rpm. The assessment is performed with points arranged circumferentially around the diffuser and evaluated at two different radial stations, $r = 55.5$ mm and $72.5$ mm. The inset to the right shows measurement probe point locations on the compressor back wall.

equation for error. For any quantity $\phi$ this is defined as:

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

the subscripts denotes the specific grid refinement level. The lower value is for the finer grid and the larger is for the coarser grid. According Celik et al. (2008) the apparent order $o_a$ of the numerical approach is given by:

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

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

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

where $\epsilon_{ij}$ is the difference in $\phi$ between grids $i$ and $j$. For a grid refinement factor $r_{ij}$ of 2 then $g(o_a)$ is zero. A solution on a hypothetical infinite resolved grid can be computed by means of extrapolation:
\[ \phi_{ext} = \frac{r_{21}^{\alpha} \phi_1 - \phi_2}{r_{21}^{\alpha} - 1} \] (5.5)

which is shown with a thick gray line in Fig. 5.3. 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:

\[ err_{ext} = \left| \frac{\phi_{ext} - \phi_1}{\phi_{ext}} \right| \] (5.6)

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

\[ GCI = \frac{1.25 err_{21}}{r_{21}^{\alpha} - 1} \] (5.7)

which can be used to estimate the uncertainty of the extrapolated result. Results for the GCI and the relative errors are listed in Table 5.2.

<table>
<thead>
<tr>
<th></th>
<th>( err_{ij} )</th>
<th>( err_{ext} )</th>
<th>GCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>( r = 55.5 ) mm</td>
<td>1.7%</td>
<td>0.5%</td>
<td>0.7%</td>
</tr>
<tr>
<td>( r = 72.5 ) mm</td>
<td>0.7%</td>
<td>0.07%</td>
<td>0.09%</td>
</tr>
</tbody>
</table>

Table 5.2: Tabulation of grid convergence parameters. The data is based on the diffuser static pressure distribution at two different radial stations.

Due to low relative errors further grid refinements would therefore only have a marginal effect on the numerical accuracy in solving the governing partial differential equations.

Another important assessment is to verify that the grid adequately captures relevant frequency tonalities of interest, i.e. as associated with certain fluctuating flow structures. In Fig. 5.4 the Power Spectral Density (PSD) based on the pressure history signal is presented for different grid resolutions and compared with available experiential data. The mid frequency range for conditions close to near optimum design efficiency is captured by all grids for the chosen diffuser probe, see Paper 1 for more details. An important tonality is the blade passing frequency, which is found in the high frequency range. This feature including its higher harmonics is correctly captured. However, it is only the fine grid level that gives a reasonable match with the measured data. In the high frequency range for the fine grid there is a trend showing an energy decay slope of approximately -7/3. This could be related to Kolmogorov’s energy decay law. However, care must be taken with any such conclusion because the theory is only valid for isotropic turbulence. Clearly, due to presence of a boundary layer, that is an assumption that may not hold for vaneless diffuser flow. Nevertheless, the limitation here is mainly due to the fact that the turbulence spectrum and the
flow conditions share the same scales. Therefore, classical turbulence theory may not work fully. For example if the blade passage is pumping energy into the inertial subrange the classical energy cascade may be disrupted.

![Figure 5.4: Power Spectral Density of pressure for the three grids. For clarification of the probe D0 location, see Fig. 5.1.](image)

Overall the result with the fine grid shows that the adopted implicit LES approach has no effect on the frequencies related to the blade passage. The effect of grid refinement is manifested in a wider resolved inertial subrange in the spectrum. As expected, the dissipation occurs mostly at the edge of the resolved scales. With improved resolution the dissipation and a given scale is reduced and thereby the spectrum gets closer to the theoretical one. Further refinement will ultimately result in no effect as the LES with better and better resolution will become a DNS. This process exhibits the approximate character of LES, which differs in principle from that of models (like the one used within the RANS framework), which do not vanish with improved resolution.

The observed differences between stable and unstable operating conditions can be expressed in terms of time-averaged velocity and Reynolds stress profiles upstream of the impeller eye, as shown in Fig. 5.5.

In the blade passages, the effect of the higher back pressure is seen to push the flow through the blade tip gap far upstream to the shroud inlet. For reduced mass-flow rates the amount of back flow increases, as quantified with the axial velocity profiles for Cases D to A in Fig. 5.5. A strong swirling motion component of the flow in the same rotational direction as the impeller is seen, which is a manifestation of the rotor angular velocity. Therefore, the entire flow upstream of the impeller swirls around the shaft axis and where a mixing takes place between the blade tip gas flow with the freshly entrained air flow. The increased mixing of flow in this local region towards off-design operating
5. Results: Highlights and Discussion

Figure 5.5: Time averaged velocity and Reynolds stress profile components in radial, tangential and axial directions. Data is extracted along a vertical line in plane P2. The profiles are normalized with the blade tip speed, $U_{ref}$. The gray horizontal lines indicate the circumferential radius at which the rotating-stall vortical structures are circulating in the P1 plane. This location is in near proximity of the inflection point. Cases A to D correspond to those marked with the same symbols in Fig. 5.2.

conditions can therefore be seen as amplified turbulent kinetic energy levels. A distinct shape of an annular separated formation can be seen in the P2 plane (shown in the $U_r/U_{ref}$ subplot in Fig. 5.5). It is manifested as radial outwards and inwards swirling flow which is a consequence due to the significant amount of back flow. It can be seen as a blockage or obstacle for the entrained air further upstream. Thus, the freshly entrained air is being funneled into the compressor with a radial inward directed flow profile (the radial velocity profiles are shown in Fig. 5.5). The back flow on the other hand is seen to spread
outwards due to the centrifugal forces but is concentrated towards the shroud side. When considering the radial and the axial velocity profiles together it is clear that an inflection point emerges for Case C, which grows for the lower mass flow rate Cases B and A. In general the flow in the inflection point may be unstable and so it is a possible onset mechanism for shear-layer instability. One effect of the shear-layer interface can be seen as an annular region with increased turbulent kinetic energy.

The thesis focuses on low-frequency flow phenomena, rotating stall and surge. Hence, there is no intention to capture the smallest scale turbulent flow features. This option is viable if the phenomena under consideration are not turbulence driven, but are results of other factors, such as system rotation and inertia. The phenomena, rotating stall and surge are subjected to orders of magnitude larger length scales but more importantly longer time scales as compared to the blade passing frequency. Consequently such features require an adequate number of time samples, with relatively low sampling rate, such that the frequencies of interest can be resolved. Therefore, a significant statistics corresponding to some 220 full impeller revolutions were accumulated for Case A and run on the fine grid. Of this approximately, 20 revolutions are needed for the early transient evolution from RANS to establish a suitable initial turbulent flow field. Therefore, data corresponding to 8 surge periods are used for statistical analysis of Case A. Since the low frequency tonality corresponding to surge was not expected for Case B, C and D, respectively, a fewer number of impeller revolutions were considered, but corresponding to 2 surge periods. One may note the fact that the phenomenon under consideration has long length scales, which implies that the effects of turbulence on surge and rotating stall are small.

The corresponding Power Spectral Density (PSD) for the audible frequency range is presented in Fig. 5.6. Spectra at probe point IS1 is assessed along one speed-line and the purpose is to evaluate differences as conditions go from near maximum design condition Case D to off-design condition at Case A. For Case D the major tonalities that are distinct and clear are found at RO = 1, RO = 5 and RO = 10. Here the blade passing frequency (BPF) equals RO = 10, since the impeller has ten blades. Elsewhere the spectrum is broadbanded. As the mass flow rate is restricted in Case C, no spectacular change is seen, except perhaps, a small bump appearing for a frequency range around half the angular velocity of the shaft (i.e. RO = 0.5). For Case B, the mass flow rate is even more restricted. Suddenly a notable low frequency tonality occurs at RO = 0.5, which is identified as rotating stall. In the previous Case C and Case D such feature cannot be distinguished. For Case A, yet another low frequency tonality is manifested at a fraction of the rotating order (43 Hz). This peak is also accompanied by one higher harmonic in the numerical data. One interesting observation for this particular centrifugal compressor is that the rotating stall tonality is still present under deep surge conditions. However, for Case A the spectra indicate that the rotating stall feature operates over a
Figure 5.6: Power Spectral Density (PSD) for Cases A to D. The data is based on the pressure time history in probe location IS1 (see Fig. 5.1 for orientation purposes). Experimental data (Gancedo et al. 2016) is added as a reference. The Rotating Order (RO) on the abscissa is a frequency normalized with the angular velocity of the impeller shaft. A one full impeller wheel rotation corresponds to RO = 1.

wider frequency range, i.e. gaining a hump characteristic rather than a sharp tonality. Based on the computed result the crucial factor leading to surge is the global response due to flow recirculation causing the emptying and filling process. To quantify the compression system it is therefore needed to look at the phase locked average incidence angle variation at leading and trailing edges of the impeller blades. The motivation is that flow incidence can be seen to influence the impellers ability to push fluid downstream and hence build up boost pressure. A quantitative representation of cycle phase averaged flow angles, flow momentum and total-to-total pressure ratio, respectively, are therefore presented in Fig. 5.7. The effect of large-scale flow structures on the blade loading term can be analyzed by filtering the flow field and blade loading term at a particular frequency. Figure 5.8 shows the phase angle evolution Φ, of the reconstructed acoustic blade loading term (top row) for off-design operating condition at the low surge frequency tonality. This figure is therefore a good complement to Fig. 5.7, since the connection between flow and acoustics
is included. It shows how the coherent flow structure in surge affect the blade loading term, which in turn affects the momentum flow into the radial diffuser for one complete surge cycle.

By phase angle $\Phi = 0$ the streamlines indicate that the flow is being pushed mainly downstream, see Fig. 5.8. Thus, the turbulent kinetic energy level reaches a minimum level. However, the back pressure is at a maximum, which induces more recirculation in the volute region, see Fig. 5.7. As an effect the flow incidence angles therefore increases while the upstream and downstream flow momentum decreases. Since the back pressure is at its peak level the corresponding blade loading distribution shows a maximum towards the rear part of the impeller surface. However, referring to a downstream directed flow is not completely satisfactory since the streamlines close to the blade suction side describes a circulating motion. There, the streamlines are seen to spiral outwards initially prior discharge. Qualitatively, this feature outlines an emerging boundary layer separation, and is associated with a strong adverse pressure gradient. By phase angle $\Phi = \pi/2$ later, all seeded streamlines in Fig. 5.8 indicate upstream direction. This means that the compressor is entering the emptying phase of the surge cycle and the flow is completely reversed. From this point in the phase cycle the pressure reduces rapidly. The turbulent kinetic energy manifests a strong gradient in the blade passages. A consequence of complete flow reversal is that the impeller cannot produced any observable downstream momentum (Fig. 5.7 at $\Phi = \pi/2$).

In this condition the impeller cannot retain its efficiency. A phase shift of $\pi/2$ is observed between the pressure and the flow momentum indicating a resonance phenomenon. By phase angle $\Phi = \pi$ the turbulence level has relaxed yet the seeded streamlines depict complete flow reversal. However, the upstream streamlines are almost parallel with the inducer plane. Therefore, the compressor is gradually undergoing a recovery from its minimum pressure level of the mode cycle.

At $\Phi = 3\pi/2$ the compressor is subjected to the filling phase of surge. The corresponding blade loading term depicts a neutral level. By phase angle $\Phi = 3\pi/2$ the compressor is recovering where the flow is now being directed downstream again. Therefore, the flow incidence angles reach a minimum and subsequently the flow momentum transferred downstream reaches high values (Fig. 5.7 at $\Phi = 3\pi/2$). The flow mode perturbation transition from $\Phi = 3\pi/2$ to $\Phi = 2\pi$ can be seen as an adjustment back to the phase evolution described at $\Phi = 0$. From this point the phase evolution will repeat in a surge limit cycling procedure. The cyclic behavior of the blade loading pressure is here seen as the acoustic source mechanism responsible for amplified noise levels in off-design operating conditions.

Another important low frequency phenomena observed in the computed LES data is associated with rotating stall. Figure 5.9 shows the phase angle evolution, $\Phi$, of the flow perturbation and the blade loading term contribution at $RO = 0.5$. 
5. Results: Highlights and Discussion

Figure 5.7: Impeller performance quantity characteristics are presented for Cases A to B. The data is phase cycle averaged based on the dominant frequency for each case: Case A, surge; Case B and C, rotating stall; Case D, time-averaged. Flow angles and flow momentum at the inducer and exducer including the total-to-total pressure ratio are circumferentially averaged at 50% blade span. The inducer blade angle is $\beta_{1b} = 60^\circ$ and the blade back sweep angle is $\beta_{2b} = 40^\circ$. These angles are defined relative to the meridional, see sketch in the upper right corner.

It should be noted that the rotating reference frame rotates with RO = 1, which is clockwise in Fig. 5.9. An observer in the acoustic farfield is located in an absolute frame of reference, which does not move. Thus, it is vital to transform the observations between moving and non-moving reference frames. The flow quantities pressure and velocity in the impeller region are sampled in the rotating reference frame in cylindrical coordinates. For an interpretation in a static (absolute) frame they should be rotated in the other direction by angular velocity RO = -1. In rotating stall a stationary observers would see the upstream vortices rotate at RO = 0.5. Hence, RO = 0.5 (absolute) equals RO = 1.5 (rotating) and subtracting RO = 1 reference frame mode. In theory, there could be a right running and a left running mode contributing to the RO = 0.5 (absolute). It may be argued that the acoustic observer cannot distinguish
5. Results: Highlights and Discussion

\[ \frac{\overline{r_L}}{0.5 \rho_{ref} U_{ref}^2} \cdot 10^{-2} \]

\[ -5.00 \quad 0.00 \quad 5.00 \]

\[ \Phi = 0 \quad \Phi = \pi/2 \quad \Phi = \pi \quad \Phi = 3\pi/2 \]

Figure 5.8: Acoustic blade loading term (top row) and the modal flow perturbation (bottom row) at the surge frequency \( RO = 0.04 \) for off-design condition Case A. The sequence from left to right shows the phase evolution of the surge mode shape.

The diagrams illustrate the blade loading and flow perturbation during the surge mode at different phases. The top row shows the blade loading term, while the bottom row displays the modal flow perturbation. Each row contains images at different phases, indicating the evolution of the surge mode.

The left running mode \( RO = -0.5 \) (absolute) is negligible compared to the right running mode \( RO = 0.5 \) (absolute). This is due to the presence of rotating stall, which is more significant in the right running mode.

In the P1 plane upstream of the impeller, circulating vortical structures can be observed, which have been identified as rotating stall. These vortical structures are cropped by the blades causing high amplitude contributions to the blade loading term at the leading edge and pressure side of the blade. This highlights the importance of understanding these flow phenomena for noise reduction and efficiency improvement in centrifugal compressors.
5. Results: Highlights and Discussion

\[
\frac{p'_{L}}{0.5\rho_{ref}U_{ref}^2} \cdot 10^{-2}
\]

\[
-2.00 \quad 0.00 \quad 2.00
\]

Figure 5.9: The top row shows the phase evolution sequence of the acoustic blade loading term. The bottom row depicts the model flow perturbation, represented with streamlines and turbulence intensity. Both sequences are synchronized from left to right and evolve around the rotating stall frequency $RO = 0.5$. The data is for Case B, i.e. near surge operating conditions.

can be observed by the high magnitude noise sources shown for all phase angles. Depending on the directionality of the vortex rotation, the flow incidence angles at the blades and hence, their loading is influenced. This causes corresponding unsteady pressures at the back wall of the impeller. At zero phase angle, $\Phi = 0$, high-pressure perturbations can be observed in two local zones at 12 and 6 o’clock near the leading edges. In between, two low-pressure perturbation zone can be found. This blade loading distribution corresponds to the two co-rotating vortex pairs found nearby just upstream in the P1 plane. The co-rotating vortex pair depicted on the vertical at $\Phi = 0$ have streamlines spinning outwards. Therefore, they are subjected to and arranged with the two elevated pressure zones on the blade surface. For the co-rotating vortex pair aligned horizontally at phase angle $\Phi = 0$ depict streamlines with inward spin. Thus, they correspond
to vortical spiraling around low pressures. This correlates with the blade loading distribution, i.e. the two low pressure zones aligned horizontally. From Fig. 5.9 it is also clear that a similar distribution resides towards the blade trailing edges at the same phase angle. However, at the exducer the distribution of the two high and two low pressures zones, respectively, are not perfectly symmetric. One of the low pressure zones is seen to dwell toward 5 o’clock for other phase angles, whereas the other low pressure zone on the opposite side is clearly convected around circumferentially. The presence of the asymmetry is one possible motivation of the subharmonic tonality found at \( \text{RO} = 0.25 \) in the power spectra density (see Fig. 5.6). This asymmetry can also be related to the asymmetric time averaged pressure field at off-design operating conditions. Due to this pressure perturbation distribution, the flow is discharged with according to these perturbations into the radial diffuser. The streamlines indicate the direction of the flow perturbations, where a radial outwards and inwards flow in the diffuser can be related to the locations of high and low pressure perturbation zones at the exducer. A phase angle \( \Phi = \pi/2 \) later in the rotating stall cycle, the impeller surface is subsequently rotated the same amount in the clockwise direction. It is important to recognize that the pressure distribution only rotate half as much, i.e. \( \pi/4 \) also in the clockwise direction. Thus, for one full impeller revolution the rotating stall feature with its blade loading signature only rotate half a revolution. This spinning motion in the clockwise azimuthal direction (i.e. absolute frame) explains the \( \text{RO} = 0.5 \) tonality found in Fig. 5.6. By a phase angle \( \Phi \) of \( \pi \) further in the cycle, the two high pressure zones near the blade leading edges are seen to be rotated into a horizontal orientation. Further on until phase angle \( \Phi = 3\pi/2 \) they are therefore seen aligned diagonally, i.e. between 10 and 4 o’clock. From this phase angle instant until \( 2\pi \) both blade loading distributions and the flow field distribution depicted with streamlines in Fig. 5.9 will gradually come back to the starting point at \( \Phi = 0 \). Once back to the starting point, the rotating stall will repeat in another cycle.

Another research highlight emerged from assessment of two-point cross correlation along a circumferential line around the middle of the diffuser channel, see Fig. 5.10. The practicality of the method is to compute coherence of the flow fluctuations. The data based on pressure for Cases B and D shows several inclined diagonal disturbances. There is one line for each fluctuation disturbance due to the blade passing. The slope of the incline indicates the group speed of the disturbance and is found to be a little higher than the speed of sound at ambient conditions. This is because the group speed gains an additional contribution from the convected flow velocity. For Cases A and B, i.e. towards low mass flow rate operating conditions, the two-point correlation (i.e. Fig. 5.10 based on circumferential velocity) is modulated by a low-frequency components, which is due to rotating stall. The effect of the rotating stall is seen to yield two disturbances convected around the diffuser for Case A and B. They correspond to two crests where the correlation is at a maximum. In between there are valleys, i.e. maximum correlation but with a negative sign. It is important to acknowledge the circumferential variation, which suggests that the rotating
stall depends on the clockwise orientation. For instance, between positions 12 o’clock and under the volute tongue the rotating stall incline is clear, indicating a speeding up behavior. After the volute tongue an apparent slowing down process is seen. For angles on the opposite side of the compressor, i.e. towards 8 o’clock, the characteristics are more diffuse showing possible checker-board pattern. Such feature can be due to the flow being locally subjected to static stall locally with back and forth motion rather than rotational. For the most restricted mass flow Case A, the characteristics based on pressure correlation display horizontal lines. Since those lines do not show any dependence in time they are immediately recognized as a pattern due to a standing wave and therefore representing surge operating conditions.

It is well known that centrifugal compressors are subjected to significant hydrodynamic pressure fluctuation levels, c.f. Gonzalez et al. (2003); Broatch et al. (2015). It is located mainly in the impeller region and they are inherently of higher magnitudes as compared to acoustic sound waves. From the compressor impeller, which is influenced by significant hydrodynamic structures, there will be a decay behavior where the averaged pressure fluctuation level reduces. From classical acoustic theory, and also supported with the LES data (Paper 3 and 4), it can be shown that the decay rate in the near-field is inverse proportional to the square of the distance from the source. For larger distances the influence from the hydrodynamic structures diminishes, and the pressure fluctuations behave inverse proportional to the distance from the source. Ultimately, there is a notable change in decay rate, which can be used to distinguish the nearfield from the farfield flow regimes. For the investigated centrifugal compressor this is located close to the bell mouth entrance, which hence defines the area where the acoustical sound field begins.

According to the Lighthills acoustic analogy and assuming flow pulsation in a straight pipe it can be show that the radiated power scales as:

$$\bar{W}_m \propto \rho_0 f^2 U^2 D^2 \frac{c_0}{\rho_0} \approx 10^{-4} \rho_0 M U^3 D^2$$

where $\rho_0$ - air density, $f$ - pulsation frequency, $U$ - mean flow speed, $D$ - pipe diameter, $M$ - Mach number, and $c_0$ - speed of sound, c.f. Åbom (2006). In order to obtain the last expression it is here assumed that the frequency $f$ is close to the surge frequency. It can be roughly estimated as one hundredth of the rotating order (RO). A source that behaves as a dipole, in centrifugal compressor applications, is due due to the rotating blade forces. For this term the frequency $f$ is assumed to be equal with the rotating order. Subsequently, the dipole power scales as:

$$\bar{W}_d \propto \rho_0 M^3 U^3 D^2$$

Viscous forces are known to influence the damping of sound waves. They emerge from the Lighthill stress tensor and are associated with quadrupole fields. Lighthill showed that the quadrupole radiated power scales as:

$$\bar{W}_q \propto \rho_0 M^5 U^3 D^2$$
Putting the result from the monopole, dipole and quadrupole power terms, respectively, together shows the following relationship:

\[ \overline{W}_m : \overline{W}_d : \overline{W}_q \propto 10^{-4} : M^2 : M^4 \]  

(5.11)

For low-subsonic flows, it can be concluded that the radiated power from the dipole radiation term is several orders more significant as compared to monopole and quadrupole terms. It must be realized that the result obtained from Lighthill’s acoustic analogy is only possible with introduction of strong assumptions. Care must be taken to adopt the result for centrifugal compressors. To clarify the relative importance on the acoustic intensity level from the
individual source terms, the spatial distribution are shown in the vicinity of the impeller, see Fig. 5.11.

\[ p'_L, \text{ scale } = 0 - 0.1 \]
\[ p'_T, \text{ scale } = 0 - 1e-6 \]
\[ p'_Q, \text{ scale } = 0 - 5e-5 \]

Figure 5.11: Acoustic intensity level distribution for off-design operating condition Case A. Surface contributions from the blade loading \( p'_L \) and the thickness \( p'_T \) source terms are shown on the impeller wall boundary. The quadrupole \( p'_Q \) is a volume source term and presented on the 50% blade span section.

It is observed that the computed blade loading term, \( p'_L \), contributes to most of the sound pressure level by an order of magnitude compared to the thickness and quadrupole terms. It is induced by the impeller and represents the interaction with flow structures primarily found at the leading and trailing edges. There, the blade loading term manifests elevated levels due to the cropping up of unsteady flow structures. The thickness term, \( p'_T \), is more pronounced towards the impeller blade tips, which is due to the higher rotational surface velocity. It is known from Raitor & Neise (2008) that tip clearance noise increases as the tip-speed approaches the sonic speed. For the considered subsonic flow regime in Case A, the influence from \( p'_T \) is thus small and even seen to be the least influential. The last acoustic source term considered in Fig. 5.11 is the quadrupole term \( p'_Q \). It is associated with turbulent fluctuations in the volume flow causing acoustic noise propagation. Elevated intensity magnitudes are observed located in the wake flow zone towards the exducer. The generation mechanism may be associated with strong gradients and large fluctuations, which are associated with the sound generation. The amplified intensity coincide with the mixing zone between the separated boundary layer flow towards the blade suction side and with the higher jet like flow zone towards the pressure side.
Chapter 6

Conclusions

The LES approach has been employed with the purpose to predict the unsteady flow field in a ported shroud centrifugal compressor equipped with a vaneless diffuser. Boundary conditions and specifics related to the numerical setup were chosen to be close replications of the cold gas-stand experimental setup at University of Cincinnati. Hence, direct comparison has been possible in terms of global performance parameters but also in terms of flow variables (time-averaged pressure and velocity as well as pressure based spectra). The computed results were found to be in good overall agreement with measured data for a range of operating conditions. For the considered geometry in this study the mass flow rate was gradually reduced from design (near optimum efficiency condition) to off-design operating conditions with the purpose to provoke low-frequency instability tonalities. Narrowband tonalities emerged in the low frequency range of the pressure spectra associated with rotating stall and surge. These modes were assessed by means of flow mode decomposition.

6.1. Contributions

It was found that a growing reversed swirling back-flow emerges as the operating conditions approach the unstable range. This is in agreement and complements the investigations made by other researchers, c.f. Guillou et al. (2012) and Hellström et al. (2012). The main reason for the back-flow can be attributed to the high back-pressures at low mass-flow rate. It has been demonstrated that the swirling annular back-flow is intrinsically linked with the alteration of incidence flow angles. It influences the rotor efficiency due to the reduced entrainment flow momentum. For more extreme conditions (even lower mass flow rates) the deep surge instability is triggered. In such scenario both downstream and upstream flow momentum transfer are seen to oscillate in a limit cycle loop, which manifest in total pressure ratio pulsation between compressor inlet and pipe discharge.

The surge limit cycle behavior was found to be linked with a phase shift between the pressure ratio and the flow momentum. In detail the peak pressure ratio is phase shifted 90° ahead of when the flow momentum approaches its minimum level. Additionally, it was also found that the incidence flow angle is synchronized with the flow momentum. For example, a large incidence
angle corresponds to a minimum flow momentum where the pressure ratio is transitioning towards a minimum level in the surge cycle. As the back pressure decreases, the compressor gradually recovers and is able to gradually buildup the flow momentum again. However, at some critical pressure ratio the momentum will gradually drop and the surge limit will repeat in another cycle.

The back-flow streaming over the blade tips is seen to reach far upstream for all the unstable, low mass-flow rate operating conditions, and hence interact with the freshly entrained air flow at the inducer. A distinct interface is therefore developed yielding a strong shear-layer. This shear-layer is seen to be energized by the swirling annular back-flow emanating near the shroud walls and off the ported shroud cavities. At the inner side of the shear-layer interface itself helical vortices are seen to take shape going into spiraling trajectories around the impeller axis. It was found that tip-leakage yielding reversing back-flow at the impeller eye is intrinsically linked with occurrences of both rotating stall and surge. The rotating stall can be seen to be governed by the shear-layer strength whereas surge is more affected by the swirling motion as induced by the back-flow. Quantified Reynolds stresses upstream of the impeller eye exposed amplified levels close with the inflection point and thus connecting shear-layer strength with the mass flow rate.

Since the flow is seen to separate near the shroud wall and reversing in the upstream direction, it induces a strong shear-layer interface. It was found that streamlines curl up into two co-rotating vortex pairs, when seeded close to the interface. This is a flow instability seen to be relatively distinct in the point spectra for near surge conditions. However, for more extreme deep surge conditions the rotating stall feature is seen to fluctuate over a range of frequencies. Thus, the co-rotating vortex pair is modulated in presence of strong swirl. The narrowbanded feature of the rotating stall instability can also be traced further downstream in the radial diffuser and volute regions, respectively. Towards off-design conditions it is known that the static pressure scalar distribution in the diffuser gains an asymmetric configuration. This can be seen as a developing low pressure zone under the volute tongue, caused by the presence of a small recirculating region. The induced pressure oscillation due to the convected rotating stall cells in the diffuser are naturally interacting with the developed recirculation region (and the shear-layer) under the volute tongue region. In other words there is a modulation of the circumferential wave under the volute tongue, which influences the boost pressure.

Another outcome of the computed unsteady flow is the possible connection between identified flow instabilities and the generated acoustic sources. Aforementioned narrowband tonalities as found in the sound power level spectra for far-field points were correlated with acoustic source terms based on the Ffowcs-Williams and Hawkings (FWH) formulation. It was found that the blade loading term is several orders of magnitude larger as compared to thickness and quadrupole terms, respectively. The blade loading source term on the impeller surface, which is dipole in character, has been quantified and exposed
at frequencies of interest, using the mode decomposition techniques. When the acoustic power from blade loading and quadrupole terms are compared it was found that their respective scaling indicates a ratio one over the Mach number squared (i.e. \( W_d : W_q \sim 1 : M^2 \)). This appears to hold in the subsonic Mach number flow regime. A similarity can be seen with the theoretical power scaling law for pulsating pipe flow via Lighthill’s acoustic analogy. It was shown that the monopole term is insignificant for the emitted sound for the considered application. However, for higher speed-lines and at larger mass-flows, this term may become more important, as the flow of the blades is expected to reach sonic speeds, see e.g. Raitor & Neise (2008).

Since the blade loading term was shown to be the most influential sound source the associated distribution on the impeller surface was assessed by means of modal decomposition. The first clear occurrence of a rotating stall tonality in the SPL spectra was detected in the second most restricted mass flow rate considered. For this case, the RO = 0.5 mode shape described a typical dipole distribution. This acoustic signature is seen to be correlated with the co-rotating vortex pair interacting upstream of the blade leading edges. Moreover, an asymmetry pressure loading resides towards the rear of the impeller surface. A higher momentum flow is manifested towards 5 o’clock acting as a low pressure sink. This region becomes amplified towards surge operating conditions. It can explain why the rotating stall tonality gains a hump feature in the PSD spectrum and provokes the subharmonic RO = 0.25 tonality.

The SPL spectrum in the upstream far-field for the most restricted mass flow rate considered expose a distinct tonality at the surge frequency and it correlates with a coherent blade loading feature distributed on the impeller surface. Since it emerges from the blade loading term in the FW-H equation it is linked with a dipole acoustic source mechanism. However, the distribution on the impeller surface does not expose a conventional dipole feature in comparison with the mode found at the rotating order or the rotating stall mode. Nevertheless, it can be characterized with a multipole expansion consisting of two monopoles in anti-phase for each impeller blade. They are distributed symmetrically in a circumferential arrangement with focus towards the rear part of the impeller surface as well as towards the leading edges and blade tips. Overall, the distribution clarifies a possible acoustic source mechanism responsible for the amplified sound pressure level at a fraction of the rotating order frequency. Ultimately, the frequency tonality associated with the surge mode was shown to describe a filling and emptying process between the impeller inlet and pipe exit discharge.

The SPL spectra in the acoustic far-field upstream of the compressor was seen to contain a second amplified feature located in the mid frequency range. This feature is demonstrated to expose more broadbanded characteristic. However, distinct narrowbanded features are embedded at twice the rotating order unity, including higher harmonics notably at RO = 3 and RO = 4. This hump feature in the SPL spectra shares resemblance with experimental observations
(Tomita et al. 2013; Evans & Ward 2005), designated as “whoosh noise”. Competing explanations exist for the mechanism behind the “whoosh noise” feature. In e.g. Torregrosa et al. (2014) it is associated with a general broadband noise in range $1 < \text{RO} < 2$ for a similar sized compressor. In Raitor & Neise (2008) it is related to “tip clearance” noise due to flow leakage between the blade tip and shroud wall. Since the time-average pressure scalar distribution around the diffuser and volute regions manifest increasing asymmetry it influences the emitted noise. It is seen to amplify the broadbanded acoustic noise level in the order of 2-4 dB. Embedded in this broadbanded characteristic there are features associated with the rotating order and its higher harmonics. From an acoustic point of view they can be defined by dipole sources distributed circumferential around the impeller surface.

The main achievements of thesis can be summarized in the following main points:

- The low frequency flow modes associated with surge and rotating stall were captured with the adopted LES methodology.
- From flow mode decomposition using either Fourier surface spectra or Dynamic Mode Decomposition it is possible to describe Surge as a system instability. Small disturbances grow until state space variables enter a surge limit cycle.
- It was found that rotating stall can co-exist with the surge feature. However, its tonality spreads over a wider frequency interval.
- The acoustic sources caused by the unsteady pressure loads on the impeller’s surface have been correlated with the acoustic spectra calculated upstream of the compressor in the acoustic regions.
- A dominant flow feature at half of the rotating order was identified upstream of the impeller inlet face. It was found that the feature consists of two co-rotating vortex pair circulating in the same direction as the impeller rotation.
- The generation mechanism of surge was shown to be subjected to large modulation of incidence flow angles and the impact on the transferred flow momentum through the blade passages.

6.2. Proposal for future work

It has been suggested by different research groups that the tip clearance noise should manifest in a narrowband range about half of the rotating order. Early investigations of “tip clearance noise” can be traced to the axial compressor community. It is acknowledged that rotating stall may evolve differently in a centrifugal compressor, yet this definition has been adopted also for centrifugal compressors. Since the tip clearance noise is not yet fully understood it would be most valuable to determine the underlying generation mechanism. It is therefore proposed to clarify the generation mechanism that is responsible for the radiated narrowbanded noise. In other words, determine if it is related to tip clearance or not. The definition that tip clearance noise strongly correlates with
half of the rotating order originates from experimental studies carried out in the 50-70s. Researchers back then did not have access to powerful computers, and the possibility to visually exploring the entire unsteady flow field was limited. It is identified in this thesis that the coherent flow structures found upstream of the impeller face illustrate a different flow mechanism. Since detailed flow visualization by means of experimental measurements is challenging it may explain why a tonality found in the power spectra density may be interpreted as a rotating pressure signature for rotating stall. In reality though, it may be related to a different thing altogether.

It is important to realize that surge and rotating stall are dynamic evolving flow instabilities. Therefore, the assumption of a fixed speed-line and a specified mass flow rate limits the possibility to find the inception mechanism. For higher fidelity prediction of the low-frequency tonalities it is recommended to assess a dynamic evolution from design conditions towards unstable off-design conditions. This may help locate a specific operating condition for which a precursor flow mode may occur prior to rotating stall and surge.

The solver elapsed time with LES for one operating condition is in the order of days running on modern supercomputing hardware facility. This means the LES computational approach is not an attractive choice for evaluation of compressor performance in early compressor design stages. As a future direction it would be possible to consider unsteady RANS (URANS). It is believed that URANS may be capable in capturing some of the instabilities observed for unstable operating conditions. Capture of self-induced rotating stall in realistic compressor geometries on the other hand may demand special considerations with URANS, such as anisotropic turbulence and curvature correction. However, if the interest is to capture mechanisms related with the noise generation in compressors a URANS formulation may not be a method of choice (due to the impact of modeling on velocity and pressure fluctuations).

Relying on direct sound propagation with compressible LES is challenging at larger distances from radiating noise sources. It is due to having a sufficient number of grid points to resolve the wavelength of acoustical propagating sound waves. Using the FW-H equation is one option to reduce the computational cost associated with far-field sound propagation. However, in the present formulation, it is mainly suitable for compact acoustic sources. Moreover, all acoustic source surfaces and neighboring volume should be in a direct “line-of-sight” and separate from the receiver location. In most radial compressor applications the volute and shroud surfaces are effectively shielding the dominant acoustic impeller surface source, introducing interference effects. However, results presented in the thesis exposed that FW-H can give good trends as compared with the direct approach in the low to medium frequency range. However, for more accurate prediction a future direction may explore the integration routines of the acoustic source terms on the right hand side of the wave propagation equation when applied to radial compressor applications.
Another possible research direction is to assess compressor sensitivity to the upstream and downstream perturbations (e.g., Exhaust Gas Recirculation - EGR pulses, air temperature conditions, pressure pulses caused by engine breathing) and analyze the impact of such perturbations on the onset of compressor instabilities.
As part of the outcome with the current PhD study, several manuscripts were published or submitted for publication. They are reproduced in Part II of the thesis and listed as follows:


A grid dependency assessment was carried out on RANS and LES data obtained for a ported shroud centrifugal compressor. Characteristic low frequency tonalities were captured with LES. They were identified as rotating stall and surge by means of advanced post-processing techniques. The candidate (ES) performed all pre-processing, solve and post-processing. The paper was written by ES with input from BS and MM. Experimental data for validation of computed results was provided by Prof. Gutmark at University of Cincinnati. ES presented the paper at the SAE 2014 International Powertrain, Fuels & Lubrications Meeting, Birmingham UK 2014.

**Paper 2** Generation Mechanisms of Rotating Stall and Surge in Centrifugal Compressors. Sundström E., Semlitsch B. & Mihaescu M. *Accepted for publication in Flow, Turbulence and Combustion, Springer, 2017*

The aim is to correlate the flow incidence angles, momentum and pressure ratios across the impeller region during phase averaged rotating stall and surge cycles, respectively. It is quantified how these parameters evolve during the low-frequency flow instability events in a realistic centrifugal compressor geometry. ES produced all data and wrote the manuscript with input from BS and MM.


Acoustic sources in a centrifugal compressor were assessed using LES data. They were found to be narrowband features associated with the angular velocity of the impeller shaft and higher harmonics. Distinct acoustic sources were also
found at low-frequency tonalities associated with surge and rotating stall. The
effect of the acoustic sources where seen to result in planar waves propagating to
the acoustic sound field at larger upstream distances. The candidate ES carried
out all necessary steps from case setup to data analysis. ES wrote the paper
with input from BS and MM. Geometry and Experimental data for comparison
was made available thanks to Prof. Gutmark at University of Cincinnati. MM
presented the paper at the 21st AIAA/CEAS Aeroacoustics Conference, Dallas,
Texas, USA 2015.

**Paper 4 Acoustic signature of flow instabilities in radial compressors.** Sund-
ström E., Semlitsch B. & Mihaescu M. *Submitted to J. of Sound and Vibration*,
(2017)

This paper is an upgraded version of Paper 3 and intended for journal publica-
tion. The flow-acoustics coupling is further assessed by looking as the two-point
cross correlation of fluctuating disturbances. Characteristic features were re-
vealed as the mass flow rate is gradually restricted with purpose to provoke
low-frequency tonalities such as rotating stall and surge. ES produced all data
and wrote the manuscript with input from BS and MM.

**Paper 5 Evaluation of Centrifugal Compressor Performance Models using
Large Eddy Simulation Data.**
Sundström E., Kerres. B (BK) & Mihaescu M. *Processing of ASME Turbo
Expo, paper id: GT2016-57169*, (2016)

A comparison of flow loss accounting was assessed using LES and reduced
order 1D modeling for a ported shroud centrifugal compressor. ES produced
the LES data and implemented the 1D model. The paper was written by ES
with input from BK and MM.

**Paper 6 Analysis of vaneless diffuser stall instability in a centrifugal com-
pressor.** Sundström E., Mihaescu M., Giachi M. (GM), Belardini E. (BE) &

LES data was produced and analyzed to characterize the mechanism of vaneless
diffuser rotating stall instability in a centrifugal compressor. The manuscript is
an upgraded version of the ETC2017-175 conference proceeding contribution
with the same title. The calculations have been carried out by ES. The boundary
conditions have been provided by GM and BE. The manuscript has been written
by ES with input from MM and VM.

**Conferences**
Part of the work in this thesis has been presented at several conferences. The
presenting author is underlined. Although related, these peer-reviewed conference
publications are not included in this thesis.


Acknowledgements

A special thanks to my main supervisors Prof. Mihai Mihaescu and Prof. Laszlo Fuchs for the opportunity to explore the unknowns and advancing the scientific frontiers a little further. Their excellent support, guidance, and engaging discussions has been vital in directing the research into fruitful areas. I would like to thank everyone in the Fluid Physics Laboratory for the inspiring environment and for the joyful Monday seminars. I’m truly grateful for the privilege to discuss findings with so many gifted researchers. Many thanks to the research group including everyone sharing the office: Johan, Martin, Bertrand, Shyang Maw, and Fredrik. The administration staff at KTH-Mechanics: Heide, Malin, and Elisabeth are acknowledged, with regret that Malin is not among us anymore.

Naturally, I would like to attribute a special thanks to my family Ellinor, Sunniva and Jan. This was made possible thanks to your encouragement and for always believing in me and being an inspiration. I would like to express a thanks to close friends for cultivating activities in spare times. Not to forget, former colleagues at Saab Bofors Dynamics and CD-adapco are recognized for previous opportunities.

The work was conducted as part of the Competence Center for Gas Exchange (CCGEx). The research was made possible thanks to financial support from the Swedish Energy Agency (STEM), Volvo Cars, Volvo GTT, and Scania. Support was also provided by BorgWarner who shared experimental data for two different investigated compressor designs. The research would not have been possible without computing resources provided by The Swedish National Infrastructure for Computing (SNIC) via HPC2N and PDC, the Parallel Computing Center at KTH. Professor Ephraim J. Gutmark and his team at University of Cincinnati, Ohio, USA, are acknowledged for providing experimental data, which has been valuable for validation of the numerical results. Finally, I would like to direct a thanks to travel funds received from the Petersohns minne foundation and also the Jubileumsanslaget från Knut och Alice Wallenbergs stiftelse, which made conference proceeding contributions possible.
Bibliography

ÅBOM, M. 2006 An introduction to flow acoustics. TRITA-AVE 2006:04, ISSN 1651-7660, ISRN/KTH/AVE/N-06/04-SE.


Hellström, F., Gutmark, E. & Fuchs, L. 2012 Large eddy simulation of the


Part II

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

Elias Sundström\textsuperscript{1} and Bernhard Semlitsch\textsuperscript{1} and Mihai Mihaescu\textsuperscript{1}

\textsuperscript{1} The Competence Center for Gas Exchange (CCGEx), KTH Mechanics, S-100 44 Stockholm, Sweden


Numerical analysis methods are used to investigate the flow in a ported-shroud centrifugal compressor under different operating conditions, i.e. several mass flow rates at two different speed lines. A production turbocharger compressor is considered, which is widely used in the heavy automotive sector. Flow solutions obtained under steady-state and transient flow assumptions are compared with available experimental data. The steady-state Reynolds Averaged Navier-Stokes method is used to assess the overall time averaged flow and the global performance parameters. Additionally, the Large Eddy Simulation (LES) approach is employed to capture the transient flow features and the developed flow instabilities at low mass flow rates near the surge line. The aim of this study is to provide new insights on the flow instability phenomena in the compressor flow near surge. Comparison of flow solutions obtained for near-optimal efficiency and near-surge conditions are carried out. The unsteady features of the flow field are quantified by means of Fourier transformation analysis, Proper Orthogonal Decomposition and Dynamic Mode Decomposition. For a near optimal efficiency set-up the frequency spectra are broad-banded with no distinct instabilities. Close to the surge line, the spectra show a distinct surge cycle frequency, which is due to flow pulsation in the compressor. The modal flow decomposition elucidates a mode occurring at the surge frequency. The mode explains the oscillating pumping effect occurring during surge. The surface spectra contours reveal the shape of the pressure pulsation during surge and support that a pressure gradient occurs with the oscillating modes found with the modal decomposition.

\section*{Introduction}
There exist several studies assessing the three-dimensional flow in a centrifugal compressor using steady-state and transient flow solvers. Precursor to the present work, the performance of a ported-shroud centrifugal compressor was predicted numerically under an idealized installation (Hellström \textit{et al.} 2012),
where the rotating reference frame has been used to handle the rotation of the impeller. The results have been validated with experimental measurements (Guillou et al. 2010) and (Hellström et al. 2010). The work has been continued (Jyothishkumar et al. 2013) using the computationally more expensive technique; the sliding mesh technique. In Semlitsch et al. (2013), the jet-like structures originating from the ported shroud cavity at different operating conditions are analyzed. The influences of the ported shroud and non-ported shroud configuration on the flow field have been lately investigated during a design and a near surge condition (Semlitsch et al. 2014). For both, ported shroud and non-ported shroud configuration, the flow field has been compared to experimental Particle Imaging Velocimetry (PIV) data and a good agreement was shown. The assessment of the flow inside a centrifugal compressor is a challenging task when experimental methods are considered. The confinement of the geometry complicates flow visualization measurements and sophisticated setups are required to deliver high quality images. Nonetheless, the flow field has been assessed by PIV measurements (Guillou et al. 2010) and by the hydrogen bubble visualization technique (Lennemann & Howard 1968). Commonly, pressure measurements are used to characterize the compressor flow. Numerical simulations are best suited to elucidate the three-dimensional flow inside of the centrifugal compressor. The steady-state RANS formulation is the computationally least demanding approach for high Reynolds number flows. Modeling allows using a rather coarse grid and therefore, simulations can be performed on a regular workstation computer. Further, the calculations can be performed in a short period of time, which allows parametric studies to be carried out in an efficient manner. Nonetheless, this approach is based on many modeling assumptions, which can lead to significant errors. Near design operating conditions, the underlying assumptions may hold and the RANS model can give reasonable results in terms of performance parameters. However, at off design conditions, large-scale oscillations can occur in the compressor flow and the steady-state approximation does not give reliable results. Unsteady RANS (URANS) simulations may be performed in this operating condition. This introduces higher computational costs. However, physical interpretations of the flow field at off design conditions are possible (Despres et al. 2013). Nonetheless, the RANS turbulence closure requires detailed verification for each application and the model restricts the interpretability of the flow field. Further, due to its dissipative nature, it is not the proper tool to be used when flow instabilities and the generated acoustics are of interest. Ideally, an approach with less modeling restrictions is applied. However, this results in an increase in computational costs. The flow in the compressor is dictated by the geometry and the employed boundary conditions. Hence, pressure forces will be several orders of magnitude higher than forces due to the wall shear stress. Further, resolving the near wall flow resembles a substantial increase in computational work. There are several unsteady based approaches, which may be used to reduce the numerical effort. For instance Broatch et al. (2014) shows a successful Detached Eddy Simulation (DES) study for radial compressor flow, a hybrid LES-RANS approach. This
Assessment of the 3D Flow in a Centrifugal compressor

methodology relies on RANS approximations in the near wall region as well as in the bulk flow when the grid size is not fine enough to resolve small eddies and on the LES approach far away from the walls where the grid resolution is sufficiently high. Nonetheless, DES was successfully employed to investigate the acoustic noise generation of surge (Galindo et al. 2008). For the different numerical simulation approaches exist different methods to handle the rotation of the wheel. For all approaches interfaces are constructed between the rotating and the static mesh components. With the frozen rotor technique, the rotor is static and rotational forces modeling the rotation are employed at the interface between the meshes. Further, a mixing plane is introduced to complete the modeling of rotating motion. Hence, with this approach, the mesh does not move and all the rotational motion is modeled. This restricts the physical representation of the problem. For transient simulations, the sliding mesh technique is a more accurate description of the geometrical rotational motion. Thereby, the meshes slide along each other at the interface and the flow streams from one side of the interface to the other. However, the rotation is updated each time step and therefore, the connectivity at the interface has to be recalculated each time step. Hence, this approach is computationally expensive. Hence, the quality of the flow assessment by simulations is a tradeoff between computational effort and accuracy. However, parametric studies require fast methods, while fundamental investigations demand high accurate flow simulations. The current study intends to analyze and compare different methods. It has been shown that an acoustic wave develops in the outlet pipe system during compressor off design operating conditions (Hellström et al. 2012; Jyothishkumar et al. 2013). However, a low frequency peak in range 20-40 Hz was observed in experiments, which was predicted at higher frequencies in simulations [1, 4]. The volume downstream of the compressor can influence the observed surge frequency. In order to improve on the low frequency content captured with the computational approach, the main purpose of this paper is to assess the CFD methodology to achieve an improved comparison with the experiments. Further, the requirements for the spatial and temporal resolutions necessary to accurately predict the unsteady flow structures near surge are evaluated. The computational resource for the numerical approaches employed are analyzed and compared. Different post processing methods and their capabilities are presented in this study. The surface frequency spectrum is a technique that has been used in other studies to interpret the rotating stall flow structure in centrifugal compressors (Mendonça et al. 2012). Modal decomposition methods are used to identify large-scale coherent flow structures. Proper Orthogonal Decomposition (POD) and Dynamic Mode Decomposition (DMD) are commonly employed and illustrate important energetic and tonal flow features. These techniques are used to identify existence of flow phenomena, which lead to excitation of the compressor surge instabilities.
Methodology

The commercial CFD code STAR-CCM+ has been used in assessment of the three-dimensional flow. The convection and diffusion terms are approximated in finite volumes with averaged values over cell faces. The steady-state RANS method used is based on the Reynolds-averaged formulation of the Navier-Stokes equations. The SST k-omega two-equation turbulence closure is employed, assuming isotropy of turbulence with curvature correction option. Thus, the gas (air) in the compressor is being compressed at moderate temperatures and assumed to behave as a thermally perfect gas. Therefore, the gas behavior is modeled using the ideal gas law using Sutherland’s law approximation for the dynamic viscosity and thermal conductivity whereas a polynomial is used to model the temperature dependence in the specific heat. In the steady-state simulations the impeller rotation is approximated with a rotating reference frame and a mixing plane interfaces giving a circumferential flow averaging at the interface. In areas with flow reversal and separation off walls, large-scale unsteady flow is expected, and hence the steady-state approach will result in poor predictions (Wilcox 1998). However, for stable operating conditions near by the maximum efficiency, most part of the compressor flow is attached. Hence, the RANS approximation can give a converged steady-state solution in short time, capturing in reasonable limits the performance parameters of the compressor. It is expected however that steady-state RANS may lead to erroneous results when approaching the surge-line. To improve predictions of the unsteady flow, an extension is made to transient fully compressible analysis using LES. The benefit with transient approaches (such as LES) is in the low mass flow end of the operating range (near surge) due to the global oscillations occurring with surge. Nevertheless, the RANS solution is used as an initial solution for the LES approach, ultimately reducing the time needed to obtain a developed flow field. Under stable operating conditions a mass flow rate and a total temperature is specified at the inlet boundary, while a static pressure is specified at the outlet boundary of the computational domain. However, close to the surge-line, the mass flow oscillates at the inlet and during performed experiments it is regulated with a lock valve at the outlet Guillou et al. (2010). To allow for “natural” breathing of the compressor under such operating conditions, a large hemispherical domain is considered upstream of the bell-mouth intake. Non-reflective conditions are considered at the inlet boundary where a Mach number target is specified according to the intended mass flow rate. The static pressure and static temperature on the inlet cell face are adjusted using the extrapolated cell centered value from the neighboring downstream cell. A mass flow boundary condition is used at the outlet of the computational domain. This is considered as a reasonable approximation of the experimental setup. In the transient LES calculations, the impeller rotation is modeled with a Rigid Body motion and sliding interfaces between the moving impeller with stationary upstream and downstream regions. LES is characterized by that the flow field is split into large and small scales by
Assessment of the 3D Flow in a Centrifugal Compressor

LES solves a large range of scales opposing to RANS, which models all turbulent scales. With LES, only unresolved scales and their interaction with larger, resolved ones have to be modeled. The smallest turbulent scales do not depend on the geometry, exhibiting universal character and are less affected by the boundary conditions. They can be modeled using SubGrid Scale (SGS) models. There are two popular SGS models named Smagorinsky (Smagorinsky 1963) and WALE (Nicoud & Ducros 1999). Both are using model coefficients that might need tuning for a particular flow problem. However, the role of the SGS model is to dissipate enough energy in order to prevent from spurious fluctuations at the small scales. The contribution of the SGS model to the flow solution is of the same order of magnitude as the truncation error of a second order discretization scheme, for a high enough grid resolution. Hence, the inherent dissipation of the numerical solver can be used to account for the SGS dissipation. The applicability of this approach can be verified post-priori by confirming the energy decay slope and range in the energy spectra. Thereby, no explicit filter of explicit model is used and therefore, this approach is called Implicit LES (ILES). Several studies [1, 6], have shown that this approach leads to reasonable predictions for centrifugal compressor flow, when the set-up is appropriate. A coupled (density-based) flow solver has been used with the RANS and LES approaches. Implicit integration is performed with a 2nd order bounded central spatial discretization. The temporal discretization is 2nd order. The Proper Orthogonal Decomposition (POD) was introduced into fluid dynamics applications to extract the large energetic flow structures from the small-scale turbulent background flow. POD analysis can be performed in industrial applications to extract characteristic flow features (Lumley 1970; Semlitsch et al. 2014b; Sakowitz et al. 2014). A short introduction is given by Lumley (1970). With the POD method, the eigenfunctions of the spatial (or temporal) correlation matrix, constructed from a series of snapshots of the velocity vector field, are computed. Thereby, the coherent motion of the flow is extracted, where the modes are favored accordingly to their energy content. However, the drawback of the POD method is the temporal representation of the modes, since a mode is not associated exclusively to a particular frequency. Nonetheless, the Dynamic Mode Decomposition (DMD) is another technique to decompose the flow into modes associated to unique frequencies (Schmid 2010; Schmid et al. 2010; Sakowitz et al. 2013). Hence, DMD is the preferred method when tonal behavior of the flow is expected, as described in Alenius (2014). However, both, the POD and the DMD methods, can be used to elucidate the large-scale unsteady flow structures.

Geometry, computational grids, and operating conditions

A production turbocharger compressor with a ported shroud is used in the numerical analysis. Figure 1 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 \[6\]. The compressor specifications are given in Table 1.

Figure 1: Used geometry of the centrifugal compressor; ported shroud supported by four unequally spaced ribs. The locations of the pressure sensors in the ported shroud cavity and on the diffuser back-plate are indicated by colored dots. The location of plane sections are shown as black lines: P2, front and side.

<table>
<thead>
<tr>
<th>Turbocharger compatibility</th>
<th>Heavy truck engine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Engine power range</td>
<td>400 to 850 kW</td>
</tr>
<tr>
<td>Number of blades</td>
<td>10</td>
</tr>
<tr>
<td>Exducer diameter</td>
<td>88 mm</td>
</tr>
<tr>
<td>TRIM</td>
<td>56</td>
</tr>
<tr>
<td>Diffuser area ratio</td>
<td>0.57</td>
</tr>
</tbody>
</table>

Table 1: The compressor design specifications.

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. 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. Local refinements are used to resolve the flow in important regions, i.e. the impeller, the volute’s tongue, the open ports, and shroud cavity, as shown in Fig. 2. Further, refinements towards the non-slip wall 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 (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 also 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 are approximations to the measurement rig and the experimental setup at the University of Cincinnati. Thus, a perfect match with experimental measurements in Guillou et al. (2010) and Hellström et al. (2010) cannot be expected.

![Figure 2: Side and front plane sections through the coarse mesh.](image)

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 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. Three successively refined grids are used to assess the dependency of the computed flow solution onto the mesh resolution. Therefore three grids, a coarse $\sim 5 \cdot 10^6$ cells, a medium with $\sim 7 \cdot 10^6$ cells, and a fine mesh with about $9 \cdot 10^6$ cells, have been used. All grids used have five prism layers near the walls. Therefore, this mesh design relies on wall functions to approximate the velocity profile down to the wall apart from refinement areas on the impeller surface and in the open ports. Wall functions are a good substitute to resolved walls if the flow is attached, as typically on planar surface or straight pipes. Under stable conditions most of
the flow in the compressor is attached so wall functions can work reasonable well within this limit. When flow separation from curved surfaces occurs, wall functions are not ideal. The prediction of the separation point is more uncertain compared to a resolved wall. As conditions approach the surge line the flow is seen to separate in local areas. One such area is the volute’s tongue but since it is a sharp edge the separation point is relatively well defined. The boundary conditions employed are listed in Table 2. A total inlet temperature of 296 K is used for all cases, i.e. Design, Near Surge and Surge operating conditions for two different speed lines. 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>RPM</th>
<th>Mass flow (kg/s)</th>
<th>Outlet static pressure (kPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design</td>
<td>64000</td>
<td>0.27</td>
<td>156</td>
</tr>
<tr>
<td>Near Surge</td>
<td>64000</td>
<td>0.08</td>
<td>165</td>
</tr>
<tr>
<td>Surge</td>
<td>64000</td>
<td>0.05</td>
<td>165</td>
</tr>
<tr>
<td>Design</td>
<td>88000</td>
<td>0.27</td>
<td>200</td>
</tr>
<tr>
<td>Surge</td>
<td>88000</td>
<td>0.166</td>
<td>200</td>
</tr>
</tbody>
</table>

Table 2: Boundary conditions imposed based on measurements at University of Cincinnati.

**LES and RANS quantitative performance prediction**

The computed values for the pressure ratio as well as the efficiency are compared quantitatively with experimental data in Fig. 3. Data is shown for the fine grid resolution set-up. The comparison is made for two different speed lines, i.e. 64000 rpm and 88000 rpm. Experimental data are displayed with dots, RANS data with rings and LES data with a cross. For the LES data the pressure ratio variation and the efficiency variation are indicated with a vertical error-bar, while the mass flow standard deviation is indicated with a horizontal error-bar. Towards the surge line, larger variations are captured due to the flow instabilities. Smaller variations are predicted near by the optimal efficiency at design operating condition.

The pressure ratio and the compressor efficiency are calculated as follows:

\[ \text{Pressure ratio, } PR = \frac{p_{02}}{p_{01}} \]

\[ \text{Efficiency, } \eta = \frac{(p_{02}/p_{01})^{(\gamma-1)/\gamma} - 1}{(T_{02}/T_{01}) - 1} \]
Figure 3: Comparison of the pressure ratio and efficiency predictions with the experimental data for different operating conditions.

where \( \gamma \) represents the specific heat ratio of 1.4. Index 0 denote stagnation pressure or temperature. The indexes 1 and 2 refer to the quantities at the inlet and the outlet, respectively. Equation (2) is an estimate of the efficiency assuming isentropic flow. It was used to estimate the efficiency based on the experimental data (Guillou et al. 2010; Hellström et al. 2010) and it is used here for a corresponding comparison. The numerical data shows a similar trend as the experimental data for a variety of operating conditions at two speed lines. Moreover, the selected LES approach exhibits a better trend towards the surge-line than the selected RANS method. The differences observed are due to both uncertainties as well as the computational accuracy. Possible uncertainties are due to:(i) the differences between boundary conditions employed during simulations and the realistic experimental conditions, (ii) the differences in geometry, (iii) absolute error in experimental data. The numerical accuracy 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 2nd order bounded central difference scheme in space.

**Steady-State RANS: Spatial Resolution Effect**

The steady-state RANS performance predictions using the three different grids, near an optimal efficiency operating condition for the 64000rpm speed line are
compared in Table 3 with experimental data. The comparison is performed in terms of Pressure Ratio (PR) and Efficiency.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>$\dot{m}$ (kg/s)</th>
<th>PR</th>
<th>$\eta$</th>
<th>Elapsed time</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>RANS</td>
<td>Exp.</td>
<td>RANS</td>
<td>Exp.</td>
</tr>
<tr>
<td>Coarse</td>
<td>0.27</td>
<td>1.55</td>
<td>1.52</td>
<td>0.72 0.70</td>
</tr>
<tr>
<td>Medium</td>
<td>1.55</td>
<td>0.72</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Fine</td>
<td>1.55</td>
<td>0.72</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 3: Steady performance prediction near an optimal efficiency condition (64000rpm). The turn-around times are based on a Cray XE6 system (2.1 GHz processors) with 24 cores per node, 32 GB RAM per node.

The variation in the global performance parameters predictions is small in between the mesh resolutions. Further, the results are in agreement with the experimentally obtained values. This suggests a small grid dependency in terms of the global performance parameters.

A grid resolution comparison of local flow parameters such as the static pressure on the volute back plate at different radial positions is also assessed. The comparison is shown in Fig. 4.

![Figure 4: Static pressure measurements on the volute back plate at two radial positions R2 (55.5 mm) and R3 (72.5 mm). Comparison of the results obtained on the Coarse, Medium and Fine grids. The location of the pressure probes in the experimental setup are indicated in Fig. 1.](image-url)
The result shows a small variation between the evaluated grid sizes. Hence, the grid sensitivity is small with respect to local flow parameters. Further, the RANS solution data follows a similar trend compared to experimental data. The higher static pressure is captured under the volute tongue and reduces at 300 degrees (see also Fig. 1 for orientation purposes). For the R2 radial position the RANS data reads a little higher than the experimental data. This might be induced due to difference in boundary conditions with the experimental data where the gap between the impeller and volute back is not included in the simulation.

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

<table>
<thead>
<tr>
<th>Part</th>
<th>Pressure (Nm)</th>
<th>Wall Shear stress (Nm)</th>
<th>Net torque (Nm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller</td>
<td>2.19</td>
<td>3e-2</td>
<td>2.2</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:

\[
\text{Power} = \text{Net torque} \cdot \omega \sim 15 \text{ kW} \tag{3}
\]

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

**LES: resolution effect**

A grid dependency study is carried out with present LES approach on three computational grids. Due to the transient character of the simulations, sensitivity of the solution to the time-step size is also assessed. The three time-steps investigated correspond to 1, 2 and 4 degrees rotation of the impeller per time-step respectively. The smaller time-step size gives a convective Courant number of about unity for the finest grid used. It is characterized by local refinement areas at leading/trailing impeller blade edges. This time-step size is expected to be the best choice for prediction of a truly unsteady flow field. However, the frequencies characterizing surge are on the order of dozens of Hz, which will require long computational spans in order to be captured. Thus, for faster turn-around, it is relevant to assess possibilities of a more aggressive time-step size, without compromising the solution accuracy. One obvious drawback with an aggressive time-step size is that too large time-steps will ultimately diffuse flow details at the sliding interfaces. The calculated pressure ratio and efficiency from the LES data are shown in Table 5.
Table 5: Unsteady performance prediction at Design condition. The turn-around time is based on a Cray XE6 system (2.1 GHz processors) with 24 cores per node, 32 GB RAM per node.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Time-step</th>
<th>PR</th>
<th>η</th>
<th>Elap.Time/rev</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>LES</td>
<td>Exp.</td>
<td>LES</td>
</tr>
<tr>
<td>Coarse</td>
<td>4°</td>
<td>1.55</td>
<td>1.52</td>
<td>0.72</td>
</tr>
<tr>
<td>Medium</td>
<td>2°</td>
<td>1.55</td>
<td>0.72</td>
<td></td>
</tr>
<tr>
<td>Fine</td>
<td>1°</td>
<td>1.55</td>
<td>0.72</td>
<td></td>
</tr>
</tbody>
</table>

The unsteady LES predicts the global performance parameters in good agreement with the experimental measurements. The turn-around time (computational time required to achieve one revolution period) for the fine mesh using the smallest time-step is an order of magnitude larger than the coarse mesh with the aggressive time-step size. To assess the quality of the LES prediction on the different grids, the time-averaged static pressure on the volute’s back-plate is compared with measurements in Fig. 5. The result shows that the LES solution data captures a similar trend compared to the experimental data. Moreover, the variation between the evaluated grid sizes is small. Hence, the grid sensitivity is small with respect to local flow parameters. The higher static pressure is captured under the volute tongue and reduces at about 300 degrees. For the R2 radial position the LES data reads a little higher than the experimental data. Thus, showing a similar trend with the RANS data as seen in the previous section. The maximum difference between the grids for the two radial positions with respect to the static pressure are located at approximately 25 and 300 degrees and is less than five percent.

Stereoscopic PIV (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 Guillou et al. (2010); Hellström et al. (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. 6 and 7. The figures show the time-averaged velocity magnitude contours obtained by SPIV measurements and LES simulation for the finest grid with 1 degree impeller rotation per time-step for Design condition at 64000 rpm.

The SPIV visualization shown in Fig. 6 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, there is a fair agreement between the LES data and the SPIV measurements in terms of the velocity magnitude levels. In Fig. 7, it can be observed that
Figure 5: Time-averaged static pressure measurements on the volute back plate at two radial positions R2 (55.5 mm) and R3 (72.5 mm). Comparison between Coarse, Medium and Fine grids with 1 deg impeller rotation per time-step for all grids.

Figure 6: Time averaged velocity magnitude contours obtained by SPIV measurement and LES simulation on the side plane section for Design condition at 64000 rpm.

the velocity magnitudes are slightly higher at the supported ribs of the ported shroud (see Fig. 1 for locations of the ribs). These velocities evolve in the jet-like structures described in [4]. The highest velocities are also measured at these locations in the SPIV measurements.

The time-history of pressure have been acquired in several probes, located as shown in Fig. 1. By calculating the Power Spectral Density (PSD) of the monitored signal the spectral content of the data can be analyzed.
The computed PSD plots from the LES data are based on a physical time range of 0.02 s corresponding to 20 revolutions. A Hanning window with six blocks and 50% overlap is used for averaging. As seen in Fig. 8 the fine mesh with small time-step size exhibits a similar trend as the experimental pressure history at the sensor location D0. Due to the relatively short physical time range in the computed PSD for the LES data the frequency content is not adequately resolved below 1000 Hz. Thus the frequency axis is clipped at this level. Although, the resolution in the low frequency range can be improved by running a larger number of revolutions, it is not needed for determining a suitable grid/time-step size for Design condition. Nevertheless, a sufficient number of revolutions is needed to capture the surge frequency which is done in the subsequent section. In the high frequency range, the pressure fluctuations are not fully captured for the medium and coarse mesh, due to the induced numerical dissipation by the growth grids and time step size. Nevertheless, all investigated mesh resolutions predict a distinct tonal peak at the blade passing frequency of 10666 Hz, where the global underlying frequency content is broadband. This points out the need for a broadband enabled turbulence model, such as LES. Whereas unsteady RANS with a two-equation turbulence model might predict the tonal blade passing frequency consistently, it is likely to underestimate the broadband frequency content, as shown by Mendonça et al. (2003).

**Surge Prediction**

From the previous analysis one can conclude that, the fine mesh and the smallest time-step size would give a reasonable consistent PSD content for frequencies up-to the blade passing frequency, but on expense of a greater computational cost. Using a coarser mesh for improved turn-around time would introduce uncertainty and hence, the ability to draw consistent conclusions. Therefore, the finest computational mesh with the small time-step size is used for the flow field prediction at surge operating conditions.

Figure 9 shows the reported mass flow rate at 0.05 kg/s for surge condition and for the speed-line of 64000 rpm. Evidently, a physical elapsed time is
needed until the surge cycle is fully developed from the initialized condition. This is seen to take approximately 0.02 seconds or 30 revolutions. The outlet mass flow is constant due to the applied constant mass flow boundary condition. Obviously, this is not the case in the experimental setup, where the mass flow rate is regulated with a lock valve until the desired operating condition is obtained. Nevertheless, with use of a free-stream non-reflective boundary condition at the inlet, the static pressure and temperature are allowed to float depending on the upstream condition and being adjusted so that a target Mach number is obtained. As a result the inlet mass flow is seen to oscillate with a characteristic surge cycle frequency. Going one step further and fitting a 1st order Fourier curve to the LES data gives a frequency of the oscillation at 43 Hz, which is in the same range as observed in the measurements (Guillou et al. 2010). The phase-averaging is computed using the frequency obtained from the Fourier curve fit giving an average spanning over eight surge cycles. It can be observed that on the underlying harmonic oscillation, a substantial amount of fluctuations with high amplitude occur.

Figure 10 shows a comparison of the frequency content monitored experimentally and numerically at the pressure sensor locations IS1 and D0. Their locations are given in Fig. 1. The measured signal is 5 seconds long (i.e. about 200 surge cycles), so that a larger number of averaging bands has been used to filter out the noise in the experimental data. In the high frequency range there are similarities in the experimental and the predicted spectra. However, in the low frequency range the LES predicts only one dominant peak frequency. The surge cycle frequency peak is captured by LES at 43 Hz. The reason for this is that the computation of the PSD from the LES data is based on much fewer surge cycles than the experimental data so those harmonics are not resolved.
The spectral signature shown in Fig. 10 for surge condition contains distinguishable narrowband and broadband features. Observable narrowband features are present at approximately 200-300 Hz and as well as in the range between 500-600 Hz.

A major peak in the spectra is the rotating order at 1066 Hz. This peak is manifested in the pressure sensor location IS1 but cannot be seen in the signal monitored at the location D0. The distinct blade passing frequency is revealed at ten times the rotating order. This is because the impeller has exactly ten blades.

**Frequency surface spectra at Surge**

The surface spectra at the surge operating condition around frequencies associated with certain flow phenomena are shown in Fig. 11 (see Fig. 1 for the locations of the three presented planes). These frequency banded surface spectra are obtained by performing Fourier analysis on the time-varying pressures in all the nodes on the surface. The signal is filtered to include a desired frequency band. The resulting SPL (dB) are plotted as a surface contour. The evaluated time series for the surface spectra correspond to approximately three and a half surge periods. Nevertheless, in Fig. 11 at the top, the spectral magnitudes around the surge frequency are illustrated, which corresponds to the mass
flow pulsation at approximately 43 Hz. The pressure ratio is building up to a certain point until the mass flow decreases at the inlet and the pressure ratio drops back to lower values again. This process repeats every surge period and this phenomenon is commonly described as the emptying and refilling of the centrifugal compressor (Semlitsch et al. 2013).

Relatively high spectral energy levels are located in the vicinity of the outlet pipe as well as off the impeller blade tips. However, at the compressor inlet rather low amplitudes are observed. In the inlet plane upstream of the impeller, the higher amplitudes form an annular shape. At the outer periphery other frequencies are dominant. In the front plane section nearby compressor’s back plate, the observed amplitudes are increasing towards the outlet. Under the tongue, a low amplitude shape is detected. In Jyothishkumar et al. (2013); Semlitsch et al. (2013) various sources of instability mechanisms causing surge are proposed. At the volute tongue region, flow was seen to separate due to an adverse pressure gradient. This shape shown in Fig. 11 in the front plane under the tongue is associated with this phenomenon. A small low magnitude contour

Figure 10: Power spectral density in pressure sensor locations IS1 and D0 for surge condition and 64000 rpm.
Figure 11: Surface spectra at surge operating condition around frequencies associated with certain flow phenomena.

can be observed. Additionally in the region of the diffuser and volute between 2 and 5 o’clock, low amplitudes are observed, which indicate a more unaffected flow stream by surge at this location. It has been shown Semlitsch et al. (2013) that higher velocities occur in this region during off design conditions. Contrary in the region on the opposite side between 9 and 12 o’clock, local increases can be observed.

In the frequency band between 240 and 260 Hz, the highest magnitudes occur under the tongue. A ring-line shape can be observed in the impeller entrance plane into the compressor. However, the magnitudes of the pressure fluctuations in the entrainment region of the compressor are low. Also in the higher frequency range between 490 and 510 Hz, which corresponds to the double of the previous frequency range, the same pattern in the entrainment region can be observed. Further, the amplitudes are focused towards the impeller wheel and decay in the exit pipe and at some distance upstream of the impeller. Close to the compressor inlet, the source of this fluctuations is suggested to steam from the interaction of back-flow, originating from the ported shroud cavity, and incoming main flow (Jyothishkumar et al. 2013; Semlitsch et al. 2013).
transient animations of the instantaneous pressure field, relatively distinct low
pressure spots are seen to spiral at the impeller inlet at approximately 250 Hz.
Moreover, the separation bubble is seen to pulsate in the volute perimeter just
downstream of the volute tongue.

At the rotating frequency of 1066 Hz, i.e. the rotating speed, the SPL (dB)
distribution is dominant in the impeller. At the blade passing frequency the
SPL 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.

Orthogonal Decomposition and Mode Decomposition at Surge

The instantaneous flow field is rather intricate to interpret due to the inherent
complexity of the flow. A lower order representation of the flow, by the large flow
structures, can be easier to interpret. The high frequency incoherent turbulent
fluctuations are filtered from the flow field.

A total of 2165 consecutive snapshots of the velocity vector field with an
interval of 26 µs between snapshots have been used for computation of the
modes. This time series corresponds to approximately two and a half surge
periods. Only the non-rotating regions have been included in the analysis.

Line integral convolution of the vector field for the POD modes are shown
in Fig. 12 on the front, side and P2 plane sections respectively.

The zeroth mode corresponds to the averaged velocity field, where the
higher modes deviate to the mean to the actual flow field. Hence, in cases the
modal shape needs to be interpreted superimposed on the other modes. The
zeroth mode shows that flow field is highly rotational in time average during
surge at the compressor inlet. A strong swirling motion can be observed, which
reduces towards the center line of the impeller. However, the mode shows an
undisturbed inflow into the compressor. Additionally, the zeroth mode shows
higher velocities in the diffuser towards the tongue. There, it can be observed
that a substantial amount of fluid recirculates from the end of the volute back
into the initial region of the volute at the tongue and only a low velocity stream
(indicating a low mass flow) leaves into the exit pipe with a strong swirling
motion.

The first POD mode characterizes the low frequency phenomena, i.e. surge,
and is shown in the mid plot of Fig. 12. Hence, the time coefficient corresponds
to a main oscillation at approximately 43 Hz. However, the time coefficient
of this POD mode contains also higher frequencies. Low mode amplitudes
and flow reversal in the mid of the compressor inlet can be observed, while
high magnitudes occur at the outer periphery of the compressor and at the
entrance into the main shroud. In the compressor inlet plane at the location of
the main shroud walls, the high magnitudes reveal a strong swirling motion,
which exhibits the same directionality as the zeroth mode. In the front plane,
an increase in the mode magnitude at the exducer towards the tongue can be
observed. Additionally, high magnitudes occur at three o’clock in the volute.
Figure 12: Surface spectra at surge operating condition around frequencies associated with certain flow phenomena. Proper Orthogonal Decomposition modes 0-2. Line Integral convolution colored with normalized vector magnitude.

The flow directivity indicated by the velocity convolution is of special interest and remarkable is that the flow in the exit pipe exhibits aligned orientated velocity convolutions towards the outlet. However, care needs to be taken interpreting the velocity convolutions, since this mode develops only superimposed onto the zeroth mode.

While the instantaneous velocity field is seen to be rather complex to quantify, the second POD mode shows a rather clear structure. Four vortical structures manifest in the plane upstream of the impeller. These structures are associated with the flow reversal from the ported shroud cavities observed
under surge conditions. One might note the influence extent of the structures generated in the inlet region, which spreads out far into the diffuser and partially into the volute.

Figure 13 shows the DMD modes 1 and 2 corresponding to the 43 Hz and 500 Hz frequency, respectively. The first two POD modes and the first two DMD modes illustrate the same behavior, and the mode shapes differ only in small details. However, the DMD modes are associated with a unique frequency. Hence, the concept of surge is easier to abstract by imagining the first DMD mode superimposed as a harmonic oscillation with the frequency of 43 Hz on the zeroth mode. The magnitude of the first DMD mode is smaller than the magnitudes in the zeroth mode.

![Figure 13: Dynamic Mode Decomposition modes 1 and 2. Line integral convolution colored with normalized vector magnitude.](image)

**Conclusions**

The possibilities of the flow field assessment in a centrifugal compressor under stable and unstable operating conditions have been investigated. The capabilities of the numerical flow solver using RANS and LES formulations have been demonstrated. A grid sensitivity analysis was presented for both simulation approaches. For the RANS model, the grid independence was proved. 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. However, 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 lead to significant differences as compared to the experimental data. In contrast to the RANS approach, the LES approach captured the trends in good agreement over the entire operating range at both speed lines investigated. 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 and the smallest time step 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 setups. However, at the high frequencies, the fluctuations were damped due to the increased numerical dissipation with large time steps. For the finest mesh with the smallest time-step an adequate match of the spectra with the experiments could be obtained.

The time to achieve an accurate solution with LES is unfortunately several orders higher than with the relatively fast RANS approach. Further, the RANS solution was used as an initial solution for the LES simulations in order to achieve faster an initial condition independent flow field. Running RANS simulations for assessing the flow in the centrifugal compressor near an optimal efficiency operating condition could 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 or tuning the two-equation turbulence model coefficient and also trying alternative models that might capture the flow curvature better.

With the LES approach, the characteristic surge frequency was captured and a much wider range of operating conditions can be investigated reliably. This 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.

The instantaneous velocity field is rather complex to quantify over time as discussed in Semlitsch et al. (2013). This paper makes use of diverse post processing method to enhance understanding of the large flow structures occurring in the compressor flow during off design operating conditions. Modal flow
decomposition methods, i.e. POD and DMD, 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 within the POD and DMD modes. Four rotating structures form at each of the ribs of the ported shroud. 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 tough, the evaluated time range in the surface spectra and modal flow analysis was based on only three and a half surge periods, the important computed modes coincide with the expected surge period. A characteristic mode corresponding to the surge phenomena was found with POD and 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 POD and DMD modes.

Acknowledgments

The authors would like to give a special thanks to The Competence Center for Gas Exchange (CCGEx), the Swedish Energy Agency (STEM), Volvo AB, Volvo Cars and SCANIA for supporting this project. Also, The Swedish National Infrastructure for Computing via HPC2N and PDC, the Parallel Computing Center at KTH, are acknowledged for providing necessary computational resources. We are also grateful to CD-adapco and their support staff. Last but not least, we are grateful to Dr. Ephraim Gutmark and Dr. Matthieu Gancedo at University of Cincinnati for sharing the experimental data used in this study.

REFERENCES


GUILLOU, E., DIMICCO, R., GUTMARK, E., MOHAMED, A. & GANCEDO, M. 2010


Paper 2
Generation Mechanisms of Rotating Stall and Surge in Centrifugal Compressors

Elias Sundström¹ and Bernhard Semlitsch² and Mihai Mihăescu¹

¹ KTH Mechanics, Competence Center for Gas Exchange (CCGEx), Osquars Backe 18. Royal Institute of Technology (KTH), Department of Mechanics, Stockholm, 10044, Sweden
² Whittle Laboratory, Department of Engineering, University of Cambridge, 1 JJ Thomson Avenue, Cambridge, CB3 0DY, United Kingdom

Accepted for publication in Flow, Turbulence and Combustion, Springer, (2017)

Flow instabilities such as Rotating Stall and Surge limit the operating range of centrifugal compressors at low mass-flow rates. Employing compressible Large Eddy Simulations (LES), their generation mechanisms are exposed. Towards low mass-flow rate operating conditions, flow reversal over the blade tips (generated by the back pressure) causes an inflection point of the inlet flow profile. There, a shear-layer generates vortical structures circulating at the compressor inlet. Traces of these flow structures are observed until far downstream in the radial diffuser. The tip leakage flow exhibits angular momentum imposed by the impeller, which deteriorates the incidence angles at the blade tips through an over imposed swirling component to the incoming flow. We show that the impeller is incapable to maintain constant efficiency at surge operating conditions due to the extreme alteration of the incidence angle. This induces unsteady flow momentum transfer downstream, which is reflected as compression wave at the compressor outlet traveling towards the impeller. There, the pressure oscillations govern the tip leakage flow and hence, the incidence angles at the impeller. When these individual self-exited processes occur in-phase, a surge limit-cycle establishes.
Nomenclature

Latin symbols
\[ \dot{m} \text{ mass-flow rate [kg/s]} \]

\[ Ma \text{ Mach number [-]} \]

\[ p \text{ pressure [Pa]} \]

\[ T \text{ temperature [K]} \]

\[ U \text{ mean velocity [m/s]} \]

\[ D_2 \text{ exducer diameter [m]} \]

Greek symbols
\[ \beta \text{ angle \[^\circ\]} \]

\[ \rho \text{ density [kg/m}^3\text{]} \]

\[ \gamma \text{ ratio of specific heats [-]} \]

\[ \eta \text{ isentropic efficiency } \frac{p_{02}^{\frac{\gamma}{\gamma-1}} - 1}{(T_{02} - 1)} \frac{T_{02}^{\frac{\gamma}{\gamma-1}} - 1}{T_{01} - 1} [-] \]

\[ \Phi \text{ phase angle [radian]} \]

Abbreviations

BPF Blade Passing Frequency

PSD Power Spectral Density

LES Large Eddy Simulation

PIV Particle Image Velocimetry

RO Rotating Order. Normalized angular velocity

Subscript

0 total quantity

1 upstream of the impeller

2 downstream of the impeller

ref variable is referred to reference value

b blade

r, θ, z radial, tangential and axial coordinates

1. Introduction

The operating range of compressors is bounded by flow instabilities at low mass-flow rates; e.g. rotating stall and surge (Paduano et al. 2001). Centrifugal compressors are commonly employed in automotive turbochargers to increase the specific efficiency of internal combustion engines (Kasseris & Heywood 2007). A wide compressor functionality range is essential to supply boost pressure at all engine operating conditions (Galindo et al. 2008). The understanding of the generation mechanisms leading to these flow instabilities would allow a more efficient flow control design, which yet puzzles researchers and engineers.

Rotating stall is a localized phenomenon restricted to the vicinity of the impeller, where stall cells propagate circumferentially across the blade passages. Different rotating stall appearances have been documented (Paduano et al. 2001; Tan et al. 2010) and control models have been developed (Lawless & Fleeter 1997). Bousquet et al. (2014) described the generation process in centrifugal compressors as a momentum balance between the incoming flow stream and the
leakage flow at the impeller blade tip, inducing reversed flow. Rotating stall has been reported to be a precursor of surge (Kämmer & Rautenberg 1982; Oakes et al. 2002), while other researches Toyama et al. (1977) observed surge without traces of rotating stall.

Fink et al. (1991) highlighted by investigating different machinery setups that surge is an instability affecting and being characterized by the entire system. They described surge as a cyclic emptying and filling process of the volume between the downstream plenum and the impeller. Based on this description, they developed a low-order model, which could reproduce the system behavior. Although the general behavior of surge has been characterized, the flow dynamics around the impeller remained yet undisclosed.

The visualization of flow structures via experimental methods is challenged by the confinement of the geometry and the violent flow fluctuations at off-design operating conditions (Krain & Hoffmann 2008; Gallier et al. 2010; Cukurel et al. 2011). Nonetheless, Particle Image Velocimetry (PIV) measurements of the ingested flow upstream of the impeller have been performed by Guillou et al. (2012), who reported strong swirling back flow at the outer perimeter at off design operating conditions. Jyothishkumar et al. (2013); Semlitsch et al. (2013) could confirm by studying the same ported shroud compressor numerically that this reversed flow originates from the tip leakage and back flow jets generated via the ported shroud. Semlitsch & Mihăescu (2016) suggested that the increased tip leakage swirling the inflow towards off design operating conditions degenerates the incidence flow angles such that the rotor blades might be incapable to maintain a constant back pressure and cause surging.

This emphasizes the practicality of unsteady computational methods such as LES for the investigation of flow structures in centrifugal compressors. Further, the potential of stability analysis via numerical methods to predict the onset of stall was shown by Sun et al. (2016), Stein et al. (2001) and Sundström et al. (2014). Vortical flow structures circulating at the impeller inlet have been visualized by Sundström et al. (2015) and Broatch et al. (2016). These, could be associated to rotating stall.

The aim of the present work is to understand the link of rotating stall and surge in centrifugal compressors and the generation of these phenomena. Therefore, we perform compressible LES calculations to investigate the flow structures at different operating points. We show that the vortical flow structures roll off the inflection point radius of the inlet profile and are therefore, liked to the tip leakage back-flow. The modes associated with these flow structures are analyzed using the modal decomposition techniques and Power Spectral Density (PSD). Further, we present the flow field at surge with a statistical analysis of the incidence flow angles, the flow field around the blades and the transferred flow momentum in order to reveal the generation mechanism of surge.
2. Geometry and Numerical Methodology

The investigated centrifugal compressor is presented in Fig. 1, where the impeller contains ten main blades with an exducer diameter $D_2 = 88$ mm, and the vaneless diffuser has an area ratio of 0.57. The compressor is equipped with a ported shroud, which is supported by four asymmetrically arranged ribs. The main objective of the ported shroud is to extend the stable operating range of the compressor map (see e.g. Yang et al. (2016)).

![Diagram of centrifugal compressor](image)

**Figure 1:** Side and front views of the centrifugal compressor CAD geometry. Locations of the pressure sensors D0 and IS1 are indicated as well as the location of planes used during data post-processing. The IS1 point is located in the middle of the ported shroud cavity, oriented 127$^\circ$ clockwise from the vertical and 0.22$D_2$ from the diffuser back wall. The impeller section intersects the impeller region at 50% blade span. The P1 and P2 planes intersect the inlet duct upstream of the impeller eye. These planes are located 0.85$D_2$ and 0.76$D_2$, respectively, upstream from the diffuser back wall. The bottom subfigure depicts inlet and outlet boundaries of the computational domain. The inset at the bottom right shows a zoomed in view of the blade passage grid resolution at 50% blade span.

Four different operating conditions are considered at a constant speedline of 64000 rpm to investigate the evolution of flow instabilities developing towards the low mass-flow rate limit. Details to the operating conditions are provided in
An idealized installation is considered to mimic the experimental setup at the University of Cincinnati (Guillou et al. 2012). Air isentrained from quiescent, standard ambient conditions via a bell mouth inlet. The hemispherical shaped inlet is located three diffuser diameters upstream from the bell mouth entrance to the compressor, where a non-reflective boundary condition treatment is applied. The region close to the compressor can be considered as a compact acoustic source field, which radiates acoustical waves in all directions towards the acoustic farfield at the speed of sound. To prevent such waves from begin reflected at the inlet boundary, i.e. back towards the compressor, the freestream velocity on the inlet cell face is corrected. The boundary normal velocity component on the cell face is obtained from characteristics, which involves extrapolation from the neighboring cell center velocity. A more detailed explanation of the non-reflecting boundary treatment can be found in the work by Giles (1990).

Correspondingly to the experimental setup, the compressor outlet duct is ten diffuser diameters long. At the outlet, the mass-flow is fixed and so is the total temperature\(^1\). Thus, the outlet pressure is allowed to float about the specific target pressure according to the experimental conditions. The walls are considered as adiabatic\(^2\) with non-slip boundary conditions.

The finite-volume compressible solver within STAR-CCM+ is utilized for the simulation of the flow. It is characterized by the integral form of the conservation equations of mass, momentum, and energy. The air flow is assumed to behave as an ideal gas, where the dynamic viscosity and thermal conductivity are assumed to be functions of the local temperature only. An implicit LES approach is adopted to reproduce the unsteady flow in the compressor.

\(^1\)The back pressure and thus the mass flow is regulated via a valve in the experiments (Guillou et al. 2012), which has been modeled by a mass flow rate boundary condition in the simulations due to the lack of the valve geometry information. The valve exhibits different impedance characteristics delaying the pressure reflection as compared to the numerical boundary condition. Therefore, surge occurs at a lower frequency in the experiments as in the simulations. Nonetheless, it can be observed that the employed numerical method captures all other tonalities.

\(^2\)This represents another difference to the experimental setup, where the compressor is driven by the turbine.
An implicit second-order upwind scheme is used for time marching. For the calculation of the convective and diffusion gradients, an implicit, bounded central third-order scheme is employed. The governing system of equations is solved as a coupled system with Gauss-Seidel relaxation in an algebraic multigrid approach. To handle the impeller rotation, the domain is divided into stationary and rotating regions, which are interfaced by the sliding mesh technique requiring information interpolation at each time step. A solid body rotation at an angular velocity of the impeller is applied to revolve the grid about the compressor axis. The computational domain is discretized by a fine unstructured polyhedral grid consisting of approximately nine million finite volumes. A low-Reynolds number wall-resolved grid with $y^+ = 1$ distribution on average is used. This is obtained with 10 prismatic cell layers, a total wall prism layer height of 0.5 mm and using a geometric stretching factor of 1.5.

The simulations have been computed for 0.2 s to converge the spectral averages corresponding to 200 rotor revolutions. More specifications of the numerical methodology have been documented in previous publications, see Sundström et al. (2014, 2015), where also a grid sensitivity study and validation with experimental data were performed. Therefore, the presently employed approach can be considered as a reliable technique to capture the flow dynamics in the compressor.

3. Results

The operating conditions under investigation are characterized by the fluctuations of the global performance parameters, as shown in Fig. 2. Small variations to the mean value can be observed for stable operating condition, while the perturbations are enhanced towards low mass-flow rate operating conditions. These are the fundamental properties of flow instabilities manifesting on the positive slope of the characteristics. With surge, the performance parameters oscillate and form a limit cycle, where the amplitude and frequency are governed by the entire system (Xinqian & Anxiong 2015).

The observed attributes at off-design operating conditions express themselves also in the flow-field, which is shown in Fig. 3. The flow velocities are lower at low mass-flow rates when the static back pressure is higher. This causes a higher residence time of the flow, manifested in the form of circulating the flow in the radial diffuser with intensified turbulent fluctuations. The turbulent kinetic energy levels increase significantly the further the mass-flow rate is reduced. The level is specifically increased at the interface between the diffuser

---

3 The disagreement can be explained by the differences in the experimental and numerical setup, as mentioned in the section “Geometry and Numerical Methodology”.

4 The limit cycle is obtained from the raw monitored system variables for Case A, i.e. $p_{02}/p_{01}(t), \eta(t), \dot{m}(t)$. These signals are first low-pass filtered and then approximated with a Fourier curve fit of 1st order. This yields the surge frequency, the phase angle, and their respective amplitudes. Since the phase angle is $90^\circ$, an ellipse shape in the phase space diagram is obtained with a semi-major axis ($p_{02}/p_{01}$ amplitude) and semi-minor axis ($\dot{m}$ amplitude).
and the volute, subject to flow unsteadiness at the trailing edge, which is similar with observations made by Lennemann & Howard (1970).

The higher back pressure pushes the flow through the blade tip gap towards the shroud inlet. This back flow, which increases the further the mass-flow rate is reduced, is quantified in Fig. 4. Due to the rotation of the rotor, the back flow exhibits inherently a swirling motion in the same rotational direction as the wheel. With the mixing process of the back flow and the freshly entrained air, the entire flow upstream of the impeller spins around the impeller axis. The high turbulent kinetic energy levels (Fig. 3) indicate the mixing of the back flow with the entrained stream upstream of the impeller, in the shroud entrance.

Towards surge operating conditions, the formation of an annular separated, radial outwards and inwards swirling flow can be clearly observed in the P1 plane shown in Fig. 3 (in terms of streamlines) and on the P2 plane in Fig. 4. The flow profiles on the P2 plane (shown in Fig. 4) reveal a significant amount of back flow, which acts as a blockage for the freshly entrained air. Therefore, the inflow is funneled causing the radial flow profile, while the back flow spreads outwards due to the centrifugal forces. Both the radial and axial velocity profiles yield an inflection point, a possible onset mechanism for shear-layer instability. Coinciding with this shear-layer interface, an annular increased region of turbulence kinetic energy can be observed in the P1 plane shown in Fig. 3.

The spectra shown in Fig. 5 exhibit tonalities on top of a broadband distributed fluctuation character. These can be related to the angular velocity
Figure 3: Flow-field characterization at the four investigated operating points in terms of time-averaged Mach number and turbulent kinetic energy. Streamlines of the velocity field are overlapped to indicate the flow direction. For the location of the post-processing planes see Fig. 1. Here, the P1 plane is offset to the right relative to the front plane.

of the impeller shaft (RO), blade passing frequency (BPF), rotating stall (RS) and surge (S). There are also frequencies related to duct resonance, where multiples (2 and 5) of the rotating order are amplified. For the stable, near
design operating condition (Case D) only the RO=1, RO=5, and the blade passing frequency RO=10 with its higher harmonic RO=20 can be observed, respectively. Additionally, a tonality in the range of approximately 30 – 50% of the rotating order unity for all low mass-flow rate operating conditions is observed\textsuperscript{5}. Due to the influence of the dominant surge oscillation at a lower frequency, the rotating stall establishes as a narrowband amplification over a finite frequency range for Case A. Further, another clearly distinct peak at RO=0.04 (43 Hz) can be observed, including one harmonic at RO=0.08. Similarly, this peak also spans a range of frequencies, which is due to the period variation over time. It suggests a complex situation with multiple period doubling until the signal may eventually come back to the same initial starting point. In between the tonalities, the spectrum is broadband. Thus the signal in those frequency ranges is stochastic with no apparent coherent flow structure.

3.1. Flow Field Decomposition revealing Rotating Stall and Surge

Modal decomposition of the sampled simulation data was performed to analyze the instabilities associated with the flow phenomena. The flow field is therefore

\textsuperscript{5}Rotating stall was observed at approximately 480 Hz, 425 Hz and 320 Hz for the Cases A-C, respectively.
\textsuperscript{6}Comparison for Case B and Case C was omitted because the experimental data exhibits non-investigated features, who’s origin is undetermined and is not visible in the numerical data.
reconstructed in terms of mean and solely the modal oscillation at the particular frequency corresponding to rotating stall (Case B) and surge (Case B). Hence, the modal decomposition is utilized to filter out the frequency of interest from the unsteady flow. The reconstructed sequence as function of its phase is shown in Fig. 6, while quantitative analysis of the impeller work is presented in Fig. 7.

At the rotating stall frequency of approximately $30 - 50\%$ of the rotating order unity (Case B), the most prominent feature is a co-rotating vortex pair being rolled off at the inner side of the shear layer formed by the back flow in the compressor inlet. Its evolution is shown in Fig. 6 in the P1 plane. Following streamlines of the phase angle sequence, it is noteworthy that the vortex pair propagates only a half revolution per rotating stall cycle. Inspecting the streamlines close to the volute tongue over a rotating stall cycle in Fig. 6, it can be observed that the flow intensely recirculates into the volute (between $\Phi = 3\pi/2$ and 0) while the flow discharges into the compressor exit pipe at other times of the cycle (between $\Phi = \pi/2$ and $\pi$). Hence, the forming shear layer

Figure 5: The Power Spectral Density (PSD) at probe location IS1 (see Fig. 1 for location orientation) is shown for all investigated operating conditions, respectively. Experimental data (Gancedo et al. 2016) is added to guide the reader. The frequency axis is normalized with the angular velocity of the impeller shaft, represented as the Rotating Order (RO). A RO=1 corresponds to one full rotation.
under the volute tongue (Semlitsch et al. 2013) is excited by the perturbations generated via the stall cells. Further, the strength of the vortex changes, which depends on its circumferential location. This is noted by the coiling up of the streamlines around the vortex cores. It is influenced by the non-axisymmetric pressure distribution in the radial diffuser at low mass flow rate operating conditions (Lorett & Gopalakrishnan 1986). It is also influenced by the back flow distribution (Sundström et al. 2017). At $\Phi = 3\pi/2$, the two streamlines on the P1 plane, seeded outside of the shear-layer, diverge radially outwards due to the centrifugal force. Streamlines seeded inside of the shear-layer eventually end up in the co-rotating vortex pair.

The manifestation of this flow instability, i.e. rotating stall, evolves similarly as described by Alekseenko et al. (1999) who analyzed a simplified scenario where helical vortices where found in a swirling flow. The radial gradient of the radial velocity component (shown in Fig. 4) generates the instability causing the vortices. The rotation of the co-rotating vortices around the impeller axis occurs due to radial gradient of the circumferential velocity component.

The essential flow feature of surge is the global response of the compression system, which is characterized by flow recirculation causing the emptying and filling process. By a phase angle of $3\pi/2$ shown in Fig. 6 for Case A, the compressor undergoes the filling phase of surge. The flow incidence angles at the leading edge of the impeller reach a minimum (as shown in Fig. 7), which coincide with similar inflow angles as observed for Case B. Therefore, the flow momentum transferred downstream reaches high values and the streamlines shown in Fig. 6, especially in the blade passages, indicate an efficient mass flow transport downstream. The high flow momentum transported downstream causes the pressure to build up in the compressor discharge pipe, which reaches a maximum at a phase angle of $2\pi$ (phase shift of $\pi/2$ compared to the momentum). The phase lag of the pressure behind the moment is evidently shown in Fig. 7. By the phase angle 0 in the surge cycle, more flow is pushed by the high pressure in the discharge pipe and recirculated in the volute underneath the tongue. By $\Phi = \pi/2$ the compressor undergoes the emptying phase, where the pressure at the discharge pipe decays rapidly. The streamlines at the inducer reverse mid-way through the blade passages with large flow separation zones, resulting in a high flow discharge angle and low flow momentum transport, as shown in Fig. 7. The two seeded streamlines on the P1 plane are subsequently being trapped with an extremely long residence time and forms multiple turns around the center axis. The impeller is not capable to retain its efficiency. Again, the pressure lags $\pi/2$ behind the low flow momentum indicating a resonance phenomenon.

As explained in the previous paragraph, the crucial factor leading to surge is the flow incidence angle at the impeller blade tip, with impact on the rotor efficiency to push fluid downstream. Figure 8 presents the modal flow perturbation at the surge frequency (case A), in terms of flow streamlines overlapped on the turbulence kinetic energy data. The data, circumferentially
Figure 6: Velocity streamlines and Mach number evolution of the surge and rotating stall modes are shown. Case A (left column) is phase averaged at the surge frequency tonality and Case B (right column) is phase averaged at the rotating stall frequency tonality. For location of post-processing planes see Fig. 1. Here, the P1 plane is offset to the right of the front plane.
averaged, reveals the variations of the incoming flow incidence angle, see e.g. cycle phase $\Phi = 3\pi/2$ as compared with cycle phase $\Phi = 0$. Moreover, the streamlines indicate large regions of flow separation off the blade surfaces towards the impeller discharge. This complements Fig. 7, which shows at surge (case A) that only very little flow momentum is transferred downstream of impeller. Thus, only a proportion of the impeller cord imparts flow momentum.

By the phase angle $\Phi = 0$ the streamlines close to the blade pressure side and in the middle of the blade passage depict downstream directed flow and the turbulence kinetic energy level is at a minimum, as shown in Fig. 8. Close to the blade suction side, the streamlines describe a circular motion (located towards the rear of the blade). They are spiraling outwards and finally are discharged downstream, into the diffuser. Qualitatively, this feature is typical
Figure 8: Phase evolution of the modal flow perturbation within the blade’s passages at the surge frequency (Case A). Flow streamlines on top of normalized turbulence kinetic energy. The data is circumferentially averaged. Between $\Phi = 3\pi/2$ and $\Phi = 0$ the incoming flow streamlines are directed downstream. From $\Phi = \pi/2$ to $\Phi = \pi$ the streamlines depict complete flow reversal for an averaged blade passage.

for an emerging boundary layer separation. By $\Phi = \pi/2$ all seeded streamlines are now going in the upstream direction depicting reversed flow of the surge mode and the turbulence kinetic energy distribution shows a strong gradient between the pressure and suction sides of the blade. The streamline seeded close to the blade suction side initially describes an outward spiraling motion before its being discharged upstream. Since the seeded streamlines are completely reversed, the impeller is not producing any notable downstream momentum (see also Fig. 7 at $\Phi = \pi/2$). Towards phase angle $\Phi = \pi$ in Fig. 8, the turbulence levels have now dropped. The seeded streamlines still depict flow reversal, but the angle at which the flow moves upstream from the impeller is almost parallel with the impeller’s inlet plane. The compressor is gradually recovering from a minimum pressure level of the mode cycle (see also Fig. 7). By cycle phase of $\Phi = 3\pi/2$ all streamlines show downstream oriented flow, apart from the streamline seeded close to the blade suction side. This correlates with flow incidence and discharge angles reaching their minimum levels, but also to that the upstream and downstream flow momentum are at their maximum levels. From $\Phi = 3\pi/2$ to $\Phi = 2\pi$ the flow mode perturbation will gradually go back to the phase evolution stated at $\Phi = 0$, and the surge limit cycle will repeat.

4. Conclusions

The unsteady compressible flow approaching surge operating conditions in a centrifugal compressor has been analyzed using LES. The numerical setup and the boundary conditions were chosen to replicate the cold gas-stand experiment
at the University of Cincinnati. Therefore, validation of the numerical predictions with experimental data was possible and documented by Sundström et al. (2014, 2015).

Focusing on flow instabilities, the flow field evolution from design to off-design conditions was assessed. Tonalities in the pressure spectra corresponding to rotating stall and surge have been shown for particular low mass-flow rate operating conditions. Flow modal decomposition analysis was employed to investigate the phenomena causing rotating stall and surge.

The observation of back flow emergence towards off-design operating conditions has been documented by many investigations, both numerically (Jyothishkumar et al. 2013) and experimentally (Gancedo et al. 2016). This back flow, streaming over the blade tips, is generated by the increased back pressures at low mass-flow rate conditions. We showed that flow incidence angles were altered by the back flow swirl, which lead to a reduction of entrained flow momentum and hence rotor efficiency. Surge was shown to be an extreme case, where the flow incidence angle and hence the downstream flow momentum transfer oscillate in a feedback loop with the build-up of back pressure in the discharge pipe causing a limit cycle.

In a surge limit cycle it was shown that the pressure ratio and the entrained flow momentum are phase shifted 90°. Thus, at peak pressure ratio, the flow momentum is 90° from reaching its minimum. The incidence angle is seen to be in phase with the momentum. When the incidence angle and the momentum reach their maximum and minimum, respectively, the pressure ratio has reduced significantly and is well on its way towards the minimum value. But since the back pressure drops it allows the flow momentum to increase and the impeller gradually recovers, and starts to produce gradually more downstream radial momentum. However, it only increases until some critical pressure ratio. Beyond this limit the momentum starts to drop again, and the surge limit cycle repeats.

At all low mass-flow rate operating conditions, the back flow over the blade tips interacts with the freshly entrained stream in the shroud entrance region, generating a strong shear-layer. In the presence and energized by the swirling back-flow the shear-layer can give rise to helical vortex formation circulating in the impeller inlet. The tonal signature of this flow instability can be traced far downstream into the radial diffuser and the volute. The asymmetric pressure distribution in the volute at off-design operating conditions results in an amplification of vortex strength towards the volute tongue. Therefore, the induced pressure oscillations are especially notable as modulation of the shear-layer under the volute tongue. Further, the perturbations induced at the impeller were shown to affect the achieved boost pressure ratio.

It was shown that the back flow at the impeller eye, manifesting due to tip-leakage, is essential for the occurrence of both phenomena, i.e. rotating stall and surge. However, the strength of the generated shear-layer in the shroud entrance governs rotating stall, whereas the induced swirl is decisive for surge. The shear-layer strength was also shown to be affected by the swirling motion of
the back-flow, as quantified with the Reynolds stresses upstream of the impeller eye.

It was found that the flow separates near the shroud wall with flow reversal going in the upstream direction, i.e. for operating conditions close to the surge line. This flow reversal induces a strong shear-layer interface. Seeded streamlines on the inner side of the interface were shown to curl up into two vortex pairs. Moreover, since a strong swirl component is present under surge condition, these vortices are seen to co-rotate in the same direction as the impeller rotation. For Case B, i.e. not in the deep surge condition, this feature is seen to be relatively distinct in the point spectrum, which means that one rotating stall period is similar with the next following one. For Case A, the rotating stall feature is seen to be more spread out over a larger frequency range. Thus, in presence of strong swirl the strength and frequency of co-rotating vortex pair are modulated. By means of seeded streamlines, it was qualitatively shown that the co-rotating vortex pair gains strength, when one of the vortices is aligned with the volute tongue.

5. Acknowledgements

This work was executed within the Competence Center for Gas Exchange (CCGEx). The work received support from the Swedish Energy Agency (STEM), Volvo Cars, Volvo GTT, and SCANIA. Computational resources were provided via the Swedish National Infrastructure for Computing (SNIC) via HPC2N and PDC, the Parallel Computing Center at KTH. Prof. Ephraim J. Gutmark and his team at University of Cincinnati provided CAD geometry and available experimental data for validation purposes.
REFERENCES


Loret, J. & Gopalakrishnan, S. 1986 Interaction between impeller and volute of pumps at off-design conditions. ASME J. Fluids Eng 108, 12–18.


Paper 3
Centrifugal Compressor: The Sound of Surge

Elias Sundström¹, Bernhard Semlitsch¹ and Mihai Mihăescu¹

¹ The Competence Center for Gas Exchange (CCGEx), KTH Mechanics, S-100 44 Stockholm, Sweden


When a centrifugal compressor operates at low mass flow rates (close to the unstable operating condition called surge), flow instabilities can develop and severe flow reversal may occur in the wheel passage. Under such conditions, noise generation has been reported resulting in a notable discomfort induced to the passengers in the cabin.

The aim with this study is to predict the flow field associated with a centrifugal compressor and characterize the acoustic near-field generation and propagation under stable and off-design (near-surge) operating conditions. The Large Eddy Simulation (LES) approach is employed. The unsteady features in the flow field leading to acoustic noise generation are quantified by means of statistical averaging, Fourier data analysis and flow mode decomposition techniques. The decomposition method is performed inside the rotating impeller region for several stable and off-design (including surge and near-surge) operating conditions. The acoustic near-field data are presented in terms of noise directivity maps and sound pressure level spectra.

For the near-surge condition an amplified broadband feature at two times the frequency of the rotating order of the shaft (possible whoosh noise) was captured. However, an amplified feature around 50% of the rotating order was captured as well. These features are present also during the investigated surge operating conditions, but occur at lower amplitudes as compared with the captured low surge frequency of 43 Hz.

1. Introduction

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) and da Silveira 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 resent years in
the automotive industry, with the adopted trends concerning engine downsizing, the provoked noise by the compressor contributes substantially to the total acoustic noise pollution. Therefore, there is an urgent need for 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 circumferential compressor. Under certain circumstances these instabilities can develop into stall cells, 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 (Fontanesi et al. 2014). However, only a few investigations analyzing the acoustic sound generation phenomena in centrifugal compressors have been conducted until recently.

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 (Rämmal & 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 narrowbanded 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 (Kameier & Neise 1997). Further, a relation between a 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 is the so-called 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; 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; 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.

Numerical simulations, opposed to experimental measurements, are well suited to visualize the three-dimensional flow inside of a centrifugal compressor, and can therefore supplement the analysis. Steady-state Reynolds-averaged Navier-Stokes (RANS) simulations in the combination with the so-called Proudman noise source model have been used for preliminary investigations (Fontanesi et al. 2014). Nonetheless, the flow at near-surge condition is of pulsating nature and consists of both broadband and narrowbanded features. Therefore, it is important to utilize a numerical approach that can capture accurately the dynamics and fluctuations associated with these flow regimes. Nonetheless, an
approach resolving all flow scales occurring in the compressor is computationally expensive and therefore 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 the more realistic results are obtained with the less restrictive modeling approach DES. With DES only the under-resolved 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.

The aim of the present study is to elucidate the differences and the occurrence of acoustic noise sources triggered by flow phenomena in the centrifugal turbocharger compressor towards off-design operating conditions. LES in combination with wall functions and the computationally expensive sliding mesh technique is employed to investigate this phenomenon, to visualize and quantify the flow instabilities in the compressor. The approach has been used previously by a limited number of research groups (Karim et al. 2013; Jyothishkumar et al. 2013; Semlitsch et al. 2013, 2014; Sundström et al. 2014).

The case setup is described in Section 2 and the numerical procedure is documented in Section 3. For a conclusive trend, three simulation cases at a low, and two at a high constant compressor speed are selected, respectively. As reference cases, the operating conditions at near optimal efficiency are simulated for each speed line. Near-surge operating conditions are simulated as well for both considered speed lines. Due to the associated computational costs, the surge operating condition is only simulated for the lower compressor speed line. The flow features of each of the cases are analyzed in Section 4 together with some of their instantaneous features. The acoustic noise radiated upstream of the compressor is analyzed in Section 5, where an emphasis is put on the possible relations to the tip clearance noise and the surge noise. A quantification of the acoustic sources and of the responsible flow phenomena is carried out using advanced post-processing techniques, such e.g. surface spectra and modal decomposition methods. The outcome is presented in Section 6 and Section 7, respectively. Finally, the results are concluded in Section 8.

2. Geometry

The centrifugal compressor under consideration (Sundström et al. 2014) is presented in Fig. 1. Both side and front views respectively of the CAD geometry are presented. The compressor is equipped with a ported shroud that is supported by four asymmetrically positioned ribs. The four narrow ports allow recirculation of the possible occurring reversal flow at low mass flow rates (off-design conditions) into the ported shroud and into the main incoming flow, as shown by Semlitsch et al. (2014). This passive flow control technique is used with purpose to widen the compressor operating range at low mass flows and high pressure ratios. The compressor is fitted with a so called bell mouth inlet which favors entrainment of still air from the ambient. This numerical set-up
replicates the experimental set-up at the University of Cincinnati (UC), see Guillou et al. (2012).

3. Methodology

The commercial Computational Fluid Dynamics (CFD) code STAR-CCM+ is used in this study. The compressible airflow is handled by solving the full compressible flow governing equations, i.e. for continuity, momentum and energy. For linkage between flow momentum and internal energy, the ideal gas equation of state is considered. Material properties are specified with dependency on local temperature variation. Sutherland’s law approximation is used for the dynamic viscosity and thermal conductivity respectively. A polynomial in temperature is considered for the specific heat. In order to capture the unsteady flow instabilities and the generated noise under unstable operating conditions, it is essential to select a broadband enabled turbulence approach. Therefore, the LES approach is adopted. No explicit subgrid scale model is considered, the approach being previously verified and validated (Sundström et al. 2014). Small eddies which are not resolved with the grid resolution are assumed to exhibit isotropic universal character. In such flow regime those eddies will dissipate into heat. The integral form of the flow governing equations, are considered and discretized according to the finite volume polyhedral grid, detailed in Fig. 1. In the previous study carried out for the same set-up (Sundström et al. 2014), the grid dependency study showed that a grid size of approximately nine million.

Figure 1: Side and front views of the centrifugal compressor CAD geometry. The locations of the pressure sensors; \( D0, \text{Pin1} \) and \( \text{Pin2} \) are indicated with a black dot. The location of planar sections are shown as black lines: front and side. The impeller has 10 blades and the enlarged inset shows a zoomed in view of the 50% blade span which indicates the local grid resolution. Ten prism layers are used to resolve the near wall flow.
polyhedral cells captured the surge period adequately. Moreover, the solution obtained showed only minor differences when compared with two coarser grids of five and seven million cells, respectively. The nine million computational grid is used in this study. The impeller has ten blades and the enlarged inset in Fig. 1 shows a zoomed in view of the computational grid at 50% blade span. At the no-slip walls on the impeller surface ten prismatic layers are considered in order to resolve the flow in the wall normal direction.

An implicit bounded central third order scheme is adopted for convective and diffusion terms and an implicit second order upwind scheme is selected for the temporal terms. All governing equations are solved as a coupled system with Gauss-Seidel relaxation in an algebraic multigrid approach. For rotation of the impeller region the rigid body motion approach is considered with sliding interfaces between stationary and rotation regions. This is considered to be a reasonable approximation for the rotation of the impeller. Adiabatic walls are assumed which means that an isentropic efficiency is considered and thereby wall heat transfer effects are neglected.

A hemispherical shaped domain is considered upstream of the compressor entrance, in order to provide conditioned boundary conditions at the inlet (Galindo et al. 2015). Thus, the hemispherical inlet surface is located about three diffuser diameters upstream from the bell mouth entrance to the compressor. This is a major difference as compared with the previous numerical studies carried out on the same compressor (Jyothishkumar et al. 2013; Semlitsch et al. 2013). In this study, to allow sound waves to propagate unhindered upstream, a non-reflective boundary condition is considered at the inlet where conditions are being adjusted to correspond to standard atmospheric conditions for still air. A Mach number target is specified to correspond to the mass flow rate. The static pressure and static temperature on the inlet cell face are adjusted using the extrapolated cell centered value from the neighboring downstream cell.

In measurements carried out for the same compressor at UC, the mass flow rate is controlled with a lock valve positioned approximately ten diffuser diameters downstream of the volute exit cone. Therefore, in the present simulations, the outlet boundary condition is extruded from the volute exit and hence positioned ten diffuser diameters downstream. Moreover, to fix the average mass flow in the system, similarly with the experimental set-up, a constant mass flow boundary condition is considered at the outlet. Pressure and temperature are extrapolated from the neighboring upstream cell centered value. With the considered extrusions the grid is expanding with a gradual coarsening towards the openings of the domain. Therefore, the grid resolution and hence the flow resolution towards the openings reduces gradually. Location of monitoring planes and probe points, used for data sampling and statistics purposes, are also included in Fig. 1.

Three simulation cases at a low (64000 rpm) and two operating conditions at a high (88000 rpm) constant compressor speed are selected for numerical
simulation, respectively. These cases have been labeled in alphabetic order according to Table 1. As reference cases, the operating conditions at near optimal efficiency are simulated for each speed line. Figure 2 shows the global performance parameters Pressure Ratio (PR) and Efficiency as calculated and as measured at UC.

<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 under investigation with the present LES.

Figure 2: Global performance parameters, pressure ratio (PR) and Efficiency from LES and Experiment for two different speed lines.

It needs to be emphasized that there are several unavoidable differences between the numerical and the experimental set-ups, mainly in terms of completeness of the boundary conditions. However, from a global performance parameter perspective Fig. 2 indicates that the simulation set-up is reasonably close with the experimental set-up and therefore capable of showing similar trends with the measurements. The vertical error bars indicating the variation in the monitored PR and Efficiency estimations, respectively. The horizontal
error bars are representing the variation in mass flow rate. Close to optimum design condition, the error bar indicates small variation which means that the flow is approximately steady. However, towards near-surge conditions the variation increases which indicates that the flow is unsteady. This motivates the need for the LES approach, which can capture large scale pulsating flow during surge condition.

4. The Flow Field

The time-averaged flow patterns and pressure data are presented in Fig. 3 for the investigated cases B, C, D, and E (see also Table 1), i.e. stable operating condition in the right column and near-surge operating condition in the left column.

Under stable operating conditions the velocity field shows that the flow enters at the bell mouth inlet and accelerates gradually towards the impeller. In the rotating impeller region, the flow gains tangential momentum and a swirl component is introduced around the center axis. In the blade passage the flow accelerates further over the blades all the way to the diffuser inlet. Thereafter, the gas expands into the larger volute volume, where it is continuously being compressed with a significant pressure gradient in the radial direction. The swirl component in the volute region is clearly visible in the front view plane just upstream of the impeller. After some residence time the flow is eventually directed towards the outlet via the volute exit cone. Under stable operating conditions the flow is overall approximately steady. With the chosen scaling for the velocity distribution and the pressure field both similarities and differences can be observed when comparing the flow fields at different operating conditions. At design condition and the higher speed-line the most noticeable difference is that the flow in the diffuser inlet and upstream of the blade passages the flow is at higher velocities, since the mass flow rate is higher. The pressure coefficient is also different to the higher pressure ratio.

When the mass flow is reduced and the operating conditions are shifted towards near-surge (the left column in Fig. 3) the streamlines on the curved plane through the blade passages shows areas where the time-averaged flow do not align with the blade surfaces, which is an indication of separated flow and stalled impeller blades. Another difference is that the steady swirl in the volute breaks up in unsteady components and this is transported further downstream via the exit cone. In the pressure field a lower pressure region is present under the volute tongue at 2 o’clock, an adverse pressure gradient is seen to develop. As a result some flow is directed towards this area and higher velocity is 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 needs to work against this stronger gradient.

As the compressor approach off-design operating condition towards the surge-line two low and two high pressure zones, respectively, are observed in the surface spectra, shown in Fig. 4. They are located opposite each other and
Figure 3: Time-averaged relative velocity distribution (normalized with the tip speed) and time-averaged pressure coefficient for flow cases B, C, D, and E as presented in Table 1, on the sections presented in Fig. 1. The relationship between the velocity in the stationary the rotating reference frame is given by $\mathbf{u}_{\text{rel}} = \mathbf{u} + \omega \times \mathbf{x}$. This means that the streamlines in the figure do not cross the sliding interface continuously. The pressure coefficient is obtained by normalizing the pressure with $\frac{1}{2} \rho_{\text{ref}} u_{\text{redC}}^2$, where the reference density is $\rho_{\text{ref}} = 1.25 \text{ kg/m}^3$. 
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 and gain strength up to the plane section $P_2$. Downstream of this plane section the large coherent vortex structure break up into smaller structures, in the vicinity of the shear flow due to the interaction with reversed flow from the ported-shroud cavities. It can also be observed that the vortex structures are not completely symmetric, which relates to the four asymmetric positioned ported-shroud ribs.

Figure 4: Instantaneous isosurface of the pressure coefficient colored with the Mach number (for Case A). The Fourier surface spectra at 50% of the rotating order ($RO = 1066$ Hz) of the pressure coefficient and the velocity distribution (on the orthogonal plane $P_2$ upstream of the impeller face) is shown to the right.

The difference between the design operating condition (with approximately steady flow) to the off-design operating condition at near-surge and surge, respectively, can be assessed by monitoring the global performance parameter $PR$ as a function of time, see Fig. 5.

The pressure ratio history for Case B consist of small periodic variations of the mass flow in the compressor, but can be considered as approximately stable, because any perturbations will decrease in amplitude with time. In the surge operating regime (Case A), the mass flow oscillates back and forth, but without net backflow at any time. A wave-like character with eight periods is observed in the surge signal at approximately 43 Hz. The sample length of the time-history considered, in both the near-surge case and the surge case is relatively short, so the signals lack resolution in the low frequency range.

Figure 6 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 $D_0$, indicated in Fig. 1. In the spectrum calculated for Case A, a strong broad peak (due to the limited number of cycles considered)
Figure 5: Pressure ratio for three different operating conditions on the 64000 rpm compressor speed line. The black curve corresponds to the design condition, flow Case C. The blue curve corresponds to the near-surge condition, flow Case B. The green curve corresponds to the surge condition, flow Case A. 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.

Figure 6: Power Spectral Density in monitor point $D_0$ from design condition to surge condition for two different speed-lines; 64000 rpm (left) and 88000 rpm (right).

appears at the surge frequency at about 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 large coherent vortex structures encountered upstream of the impeller face, which circulate 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 broadbanded 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. 7.

Case A: $\dot{m} = 0.05 \text{ kg/s}, \omega = 64000 \text{ rpm}$

Case C: $\dot{m} = 0.28 \text{ kg/s}, \omega = 64000 \text{ rpm}$

Figure 7: 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. 1) 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.

Boundary layer separation and varying upstream flow angle is observed under surge condition in comparison with design condition. A low pressure zone is visible in the pressure coefficient aligned with the volute tongue at the diffuser inlet. Two low pressure zones are visible upstream of the blades. The velocity vector field for Case A shows boundary layer separation on both the pressure and the suction side of the blades. This separation is also seen to follow the rotation. Overall, this indicates a mix of attached as well as boundary layer flow separation depending on the impellers orientation relative to the volute. The pressure fluctuations show a low pressure zone near 2 o’clock, which is aligned with the pressure gradient under the volute tongue. The opposite side at 8 o’clock, shows a higher pressure zone. These events in the flow field repeat every revolution and ultimately manifest in a tonal narrowband feature in the
frequency spectra at the rotating order. However, there are other narrowband features embedded in the signal. Looking closely at the trailing edge region of the blades (at 8 o’clock) a varying downstream flow angle is seen to enter the diffuser. This relates to developing trailing edge vortices which propagate into the diffuser and correlate with the high tonality at the blade passing frequency, observed in Fig. 6.

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.

5. Acoustical propagation

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 on the flow, the turbulence intensity is much lower. There the acoustic pressure fluctuations become distinctively visible and may be observed and quantified.

Figure 8 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 far-field. 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 distances from the source exceeding a wavelength the pressure fluctuation falls of like the inverse of the square of the distance from the source, see Fig. 9.

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 ($\text{Pin}_2$) is presented in Fig. 10. The spectra in this point show both similarities and differences compared to the spectra in point $D_0$ located in the
Figure 8: Side view section colored with $p'$ with a narrow range -0.1 to 0.1 kPa to detect sound wave propagation in the near field upstream of the bell mouth as well as OASPL (105 to 160 dB). Near-Surge Case B (1st and 3rd image) and design condition Case C (2nd and 4th image), both at 64000 rpm.

Figure 9: Root mean square of pressure fluctuation as function of distance from the impeller source along the center axis (Case C). The pressure in the far-field falls off like the inverse square of the distance from the source.

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 signal has a broadband character. Moreover, a broadband character is captured with an interesting feature at 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 lacks resolution, but can be enhanced with a more generous simulation run time and hence sample length. Nevertheless, the dominant surge peak is captured.

Figure 10: Sound Pressure Level in the hemisphere on curve R1 at $\theta = 30$ deg (side-view plane) for different flow cases, see Fig. 1 for orientation purposes.

6. Frequency Spectra

Figure 11 presents the root mean square (RMS) of the static pressure field on the centrifugal compressor wall surfaces for a selective frequency range or at frequencies which correspond to the observed instabilities in the PSD plots. This analysis is performed in order to elucidate the origin of the strong pulsation of the pressure in the compressor under near-surge operating conditions.

In the frequency range 0-10666 Hz, which represents the overall scalar field up to the blade passing frequency, strong fluctuations are localized primarily to the impeller trailing edges in the vicinity of the diffuser passage. A strong fluctuation is also seen in a small area localized under the volute tongue. The reason for the high pressure fluctuations in those areas can be explained by boundary layer separation. Therefore, the excitation of the surge period is believed to have its origin from the impeller trailing edge into the diffuser passage.

The other images in Fig. 11 show the same pressure field but at frequency values which exhibit interesting narrowband features. At the surge period at approximately 43 Hz the highest values are localized to the volute exit. The pressure ratio is building up to a certain point until the mass flow decreases at the inlet and the pressure ratio drops back to lower values again. 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
Figure 11: Side and front views, respectively, of no-slip wall surfaces colored with the RMS of Static Pressure at the surge operating condition (Case A) around frequencies associated with certain flow phenomena.

also seen under the compressor tongue. This relates with the adverse pressure gradient as observed in Fig. 3. On the left hand side of the diffuser passage from 6 o’clock to 12 o’clock strong fluctuations can also be seen. These fluctuations are related with boundary layer separation mode that can be seen in a DMD mode at the same frequency at near-surge condition as it will be described in the subsequent section.

The next narrowband feature at 250 Hz reveals strong pressure fluctuations at the leading and trailing edges of the impeller blades caused by flow separation. Similarly at this frequency the fluctuation under the volute tongue is also clear. At the narrowband feature at 500 Hz, which is a doubling of the previous narrowband feature, most of the fluctuations are localized circumferentially around the diffuser passage. At the rotating frequency of 1066 Hz, i.e. the rotating speed, the RMS distribution is dominant in the impeller.
At the blade passing frequency 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. There are also strong pressure fluctuations in the ported shroud cavities which develop into axial bands on the upstream no-slip wall surface up to the bell mouth inlet. There pressure waves are seen to propagate upstream into the hemisphere at the speed of sound.

Figure 12 shows the surface spectra of the pressure coefficient at the blade passing frequency, for Case A. The major difference in Fig. 12, as compared with Fig. 11 is that the surface spectra is presented on the sections inside the compressor, as indicated in Fig. 1, and not directly on the no-slip wall surfaces. 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 or the RO. This is due to the mass flow outlet boundary condition, which is reflective, and replicates the experimental measurement set-up at UC (Guillou et al. 2012), where the mass flow is regulated with a lock-valve.

![Figure 12: Fourier surface spectra of the pressure coefficient at the blade passing frequency (Case A). Pressure waves are observed propagating upstream into the hemisphere.](image)

7. 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 consecutive snapshots of the static pressure field on the impeller wheel surface as well as the velocity field at the 50% span impeller section with an interval of 26 µs between snapshots have been used for the
computation of the modes. These time series correspond to approximately two surge periods. The operating condition considered for this DMD analysis is the surge condition at 64000 rpm, Case A.

Figure 13: Normalized DMD magnitude, the color ranges from -1 (blue) to 1 (red). DMD modes: 1st at the Surge frequency, 2nd at the RO, 5th at 1.5RO and 6th at 2RO).

The 1st mode at the surge frequency shows 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. Moreover the field is fixed relative to the impeller rotating reference frame. In the 2nd mode at the RO 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 impellers rotating reference frame, the mode 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 2nd mode, but here the fluctuations are focused at the leading blade edges and neutral towards the rear end of the impeller surface. The 6th modes shape is at 2 times the rotating order and is thus the 1st harmonic of the RO. It is also seen to be spinning with respect to the rotating reference frame. For the mode shape number 9, the shape is
seen to be tripolar and with a frequency 3 times the rotating order. 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 14 shows the normalized DMD magnitude for the seven most energetic mode shapes (including the 0th mode which represents the mean flow). This is plotted in a bar plot with respect to both the mode number as well as the mode frequency. The subplots on the top row are DMD modes computed from the pressure field on the impeller surface whereas the bottom row are computed from the velocity vector field on the curved impeller section at 50% blade span.

The DMD magnitude from the most energetic mode is used to normalize the other modes. From the magnitude plots there is a similarity between the impeller surface and the impeller 50% span section in that there are only a handful number of modes with significant magnitudes. Beyond approximately the 7th mode the impact of higher order modes is low. The DMD computed from the velocity field at the 50% span section reveals more modes with significant magnitude in comparison with the DMD computed from static pressure on the impeller wheel surface.

Figure 15 shows the normalized magnitude of the DMD vector field (based on the velocity vector field) colored on the 50% span impeller section. The
largest magnitude from either the radial, tangential or axial relative components are used in the normalization of each mode. This in order to reveal if a particular velocity component is more significant in the mode shape.

![Normalized DMD vector field components colored on the 50% span impeller section.](image)

**Figure 15:** Normalized DMD vector field components colored on the 50% span impeller section. DMD modes: 4th at the surge frequency (Case A), 1st at RO (Case A) and 2nd at 0.5RO (Case B). The radial component in the left column, the tangential component in the middle column and the axial component in the right column.

The zeroth mode (not included) corresponds to the average velocity field. The higher order mode shapes superimposed on the mean (zeroth) mode represent the actual velocity vector field. The 4th mode shape in Fig. 15 has a frequency which is close to the surge frequency at 43 Hz. In all fairness it must be mentioned that there are a number of modes at similar low frequencies, but they show similarities in that the axial flow component is more significant compared to the radial and tangential components. Moreover, the mode shows that the axial velocity component on the mid part of the blades on both the suction side and the pressure side oscillates back and forth at the surge frequency as well as upstream at the impeller face. In other words the flow is seen to separate off the boundary on the mid part of the blades. The radial and tangential components, although being less significant than the axial component, are observed to be 90 degrees phase shifted and rotate in the clock-wise direction relative to the rotating frame of reference.
At the rotating order the radial and tangential components are seen to be more significant than the axial component and seen to oscillate in the upstream and downstream field with the middle of the blade passage being neutral. Focusing on the tangential velocity characteristics, the flow is then seen to swap direction upstream of the the leading edges and being directed by the blades all the way to the diffuser inlet station.

In flow Case B a significant mode shape is revealed at 0.5×RO, but is not manifested as a significant mode shape in flow Case A based on the evaluated velocity field on the 50% blade span section. This mode shape has an interesting axial velocity feature upstream of the blade passages. There are two jets going in the downstream direction and two neighboring and equal jets but going in the opposite direction. Since the frequency for this mode is at 50% of the rotating order, it may be concluded that this feature is related with the vortices upstream of the impeller face circulating at approximately the same frequency.

8. Conclusions

The sound pressure level point spectra in the acoustic field in the hemisphere contains 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; Semlitsch et al. 2013; 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 elucidate 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. Moreover, axial velocity fluctuation is seen to be more significant compared to the radial and tangential velocity components, respectively.
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. This correlate with the 3rd DMD mode at the near-surge condition. There two axial velocity jets are observed going in the downstream direction and two neighboring and equal jets but going in the opposite direction.

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 low pressure node 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 node 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 radial and tangential components are more significant than the axial component. On the 50% blade span section these velocity components are seen to oscillate in the upstream and downstream field with the middle of the blade passage being at neutral.

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 shows narrowband features in this frequency interval with tonality at higher harmonics to the RO. This correlates with a number of harmonic modes 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 vortices 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.

The hemispherical shaped domain considered upstream of the compressor entrance, is a major difference as compared with the previous numerical studies carried out on the same compressor (Jyothish Kumar et al. 2013; Semlitsch et al.
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.

Acknowledgments

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 is greatly appreciated. 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. Last but not least we are grateful to Dr. Gutmark and his team at University of Cincinnati for sharing the experimental data as well as the geometry.

References


Drela, M. & Youngren, H. 2008 A user’s guide to mises 2.63. As of, February .


Paper 4
Acoustic signature of flow instabilities in radial compressors

Elias Sundström¹ and Bernhard Semlitsch² and Mihai Mihaescu¹

¹ KTH Mechanics, Competence Center for Gas Exchange (CCGEx), Osquars Backe 18. Royal Institute of Technology (KTH), Department of Mechanics, Stockholm, 10044, Sweden
² Whittle Laboratory, Department of Engineering, University of Cambridge, 1 JJ Thomson Avenue, Cambridge, CB3 0DY, United Kingdom


Rotating stall and surge are flow instabilities contributing to the acoustic noise generated in centrifugal compressors at low mass flow rates. Their acoustic generation mechanisms are exposed employing compressible Large Eddy Simulations (LES). The LES data are used for calculating the dominant acoustic sources emerging at low mass flow rates. They give the inhomogeneous character of the Ffowcs Williams and Hawkings (FW-H) wave equation. The blade loading term associated with the unsteady pressure loads developed on solid surfaces (dipole in character) is found to be the major contributor to the aerodynamically generated noise at low mass flow rates. The acoustic source due to the velocity variations and compressibility effects (quadrupole in character) as well as the acoustic source caused by the displacement of the fluid due to the accelerations of the solid surfaces (monopole in character) were found to be not as dominant.

We show that the acoustic source associated with surge is generated by the pressure oscillation, which is governed by the tip leakage flow. The vortical structures of rotating stall are interacting with the impeller. These manipulate the flow incidence angles and cause thereby unsteady blade loading towards the discharge. A low-pressure sink between 4 and 6 o’clock causes halving of the perturbation frequencies at low mass flow rates operating conditions. From two point space-time cross correlation analysis based on circumferential velocity in the diffuser it was found that the rotating stall cell propagation speed increases locally in the low pressure zone under the volute tongue. It was also found that rotating stall can coexist with surge operating condition, but the feature is then seen to operate over a broader frequency interval.

1. Introduction

Downsized reciprocating internal combustion engines (ICE) in combination with turbocharging play an important role increasing the energetic efficiency
and reducing emission levels. In the wide range of engine operating conditions, the turbocharger compressor can become a very important noise contributor (Gonzalez et al. 2003). This audible discomfort is caused by the acoustic pressure fluctuations, with the origins in the unsteady pressure loads, some of which driven by the developed flow instabilities interacting with the moving or stationary solid surfaces in the compressor (Broatch et al. 2015; da Silveira Brizon & Medeiros 2012).

Without a complete description for the temporal and spatial evolution of the flow instabilities and the unsteady pressure loads on solid surfaces there is a lack of knowledge regarding the sound generation mechanism. This restricts the optimization of compressors with respect to acoustic performance (Wenzel 2006; Fontanesi et al. 2014). Rämmal & Åbom (2007) reported that such sources can be categorized according to their characteristics and are commonly referred to as monopoles, dipoles, and quadrupoles. A monopole source is associated with temporal variations caused by motion of a discontinuity forcing a net volume flow (e.g. impeller blade surface). A dipole source corresponds to pressure fluctuations on a solid surface induced by e.g. flow structure interaction with the surface or flow separation. Quadrupole sources originate from turbulent fluctuations or by varying tangential shear stresses on surfaces.

Several distinct acoustic noise features have been reported in the literature. However, a quantification of the acoustic sources has not been carried out. A distinct mode is associated with the blade passing frequency (BPF), i.e. the rotating order times the number of impeller blades. Narrowbanded tip clearance noise has been reported by Raitor & Neise (2008) to dominate over the blade passing tones. It has been reported to occur at approximately 50% of the rotating order frequency (RO) (Kameier & Neise 1997) for some compressor designs and may be more prominent at lower impeller speeds. Additionally, it has been suggested to emanate from secondary flow motion in the gap between the compressor casing and the impeller blades similar to axial compressors. Moreover Kameier & Neise (1997) hypothesized for an existence of a relationship between rotating flow instability and tip clearance noise. However, in the work by Galindo et al. (2015) the tip clearance size is reported to have negligible effect on the noise generation at off-design operating conditions. Mendonça et al. (2012) found that the occurrence of stalled impeller blades is related to a rotating instability and with associated amplified noise levels. The presence of narrowbanded noise at higher harmonics to the shaft rotational speed has been reported by Tomita et al. (2013). Another noise source, evident at near-surge line operating conditions, is coined “whoosh noise” Evans & Ward (2005); Teng & Homco (2009), stretching over several orders of kHz. It was observed that the “whoosh noise” is more apparent at near-surge operating conditions than at actual deep-surge operating conditions, see Evans & Ward (2005); Broatch et al. (2015).

Karim et al. (2013) related high flow in incident angles at the leading edges of the impeller with noise generation and suggested that rotating flow
structures in the blade passages are causing the so called “whoosh noise”. In contrast, Evans & Ward (2005) observed the main noise source to be localized further downstream in the compressor outlet piping. All these studies reveal the challenge associated with connecting the perceived acoustic noise with the actual generation mechanism. One of the best-known descriptions for noise assessment of rotating devices such as impellers is the Gutin analogy Gutin (1936). An alternative theory is according to Farassat (1981). Both models have in common that the net force produced by each blade is assumed constant. The resulting force distribution yields a source signal with a dipole character. It consists of the fundamental blade passing frequency including higher harmonics. With simplifying assumptions such as compact source field, the analogy suggests a high directionality of the farfield sound, i.e. zero intensity perpendicular and co-planar to the impeller disc. From observations, this never happens, because effects from unsteady flow disturbances will tend to manifest in a broadband spectrum. Therefore, a deterministic acoustic model is not completely satisfactory to describe the emitted noise for centrifugal compressors. An obvious shortcoming is the inability of the reduced order models to account for sound associated with low-frequency tonalities. In fact most of the audible frequency range is unaddressed. More predictive capabilities were demonstrated in Galindo et al. (2015) using unsteady Detached Eddy Simulation (DES). The computed pressure fluctuation spectra were reported to show good trends with experimental data.

The current work is looking to reveal the interconnectivity between the flow field instabilities, the acoustic sources, and the observed emitted sound levels. High-fidelity compressible Large Eddy Simulation (LES) calculations are employed in order to quantify the developing flow instabilities as the compressor operating conditions approach the surge line. This methodology, i.e. LES has also demonstrated predictive quality for a range of operating conditions as compared to probe point spectra obtained experimentally, see Sundström et al. (2014, 2015). Acoustic sources are determined using the Ffowcs Williams-Hawkings (FW-H) equations (Ffowcs Williams & Hawkings 1969). The distribution of the acoustic source signatures are extracted using modal decomposition methods. Further, the acoustic sources are analyzed and linked with the generating flow structures. From the theoretical stand point of view, c.f. Åbom (2006); Howe (1998), one can derive that radiated power due to pulsating pipe flow scales as:

\[ W_m \propto \frac{\rho_0 f^2 U^2 D^2}{c_0} \approx 10^{-4} \rho_0 M U^3 D^2 \]  

where \( \rho_0 \) - air density, \( f \) - pulsation frequency, \( U \) - mean flow speed, \( D \) - pipe diameter, \( c_0 \) - sound speed, \( M \) - Mach number, and subscript \( m \) refers to a monopole source. The last expression in Eq. 1 is obtained by assuming that the frequency \( f \) is close to the surge frequency, which is estimated as one hundredth of the rotating order frequency (RO). A one full impeller rotation corresponds to RO unity. The rotating blade forces can be represented as a dipole radiation.
For impeller blades \( f \) may be taken as the rotating order and power scales as:

\[
\overline{W}_d \propto \rho_0 M^3 U^3 D^2 \tag{2}
\]

where subscript \( d \) refers to a dipole. The Lighthill stress tensor behaves as a quadrupole acoustic source. It can be shown that the quadrupole radiated power scales as:

\[
\overline{W}_q \propto \rho_0 M^5 U^3 D^2 \tag{3}
\]

where the quadrupole is indicated with subscript \( q \). Comparing the quadrupole and dipole terms with the monopole the following relation is obtained:

\[
\overline{W}_m : \overline{W}_d : \overline{W}_q \propto 10^{-4} : M^2 : M^4 \tag{4}
\]

Consequently for subsonic flow the radiated power from the dipole term is orders or magnitude larger as compared to monopole and quadrupole terms. This result is only possible with introduction of strong assumptions. Therefore, LES together with FW-H can help to complement the theory and establish more fidelity in the acoustic power scaling law for centrifugal compressors.

Compressible Large-Eddy Simulation (LES) calculations are used to quantify the flow and the near-field acoustics associated with a centrifugal compressor at stable and off-design (unstable) operating conditions. Featured instabilities in the flow field at off-design operating conditions are linked with acoustic noise generation.

2. Geometry and Numerical Methodology

Side and front views of the investigated centrifugal compressor are presented in Fig. 1. The compressor features a ported shroud supported by four ribs in a an asymmetric arrangement. An impeller with 10 main blades (exducer diameter \( D_2 = 88 \text{ mm} \)) is considered and the compressor accommodates a vaneless diffuser. A bell mouth is fitted at the inlet and an idealized straight duct is considered at the volute discharge. The experimental data from University of Cincinnati (UC) Guillou \textit{et al.} (2012) have been used for verifying and validating the compressible solver employed. A general good agreement between predicted and measured data in terms of time-averaged velocity, static pressure distribution, frequency spectra, and compressor performance parameters (pressure ratio and efficiency) has been found Sundström \textit{et al.} (2014, 2015); Semlitsch & Mihăescu (2016).

Continuity, momentum and energy conservation constitutes the compressible flow governing equations and are solved numerically using the finite-volume compressible solver in STAR-CCM+. A compressible Newtonian fluid is assumed together with constitutive relations to couple velocity gradients to components of the stress tensor. The dynamic viscosity is thermally variant by means of the Sutherland equation. The fluid is assumed approximately calorically perfect with constant specific heats. Molecular diffusion fluxes of heat and mass are

\[\text{The back pressure and thus the mass flow is regulated via a valve in the experiments Guillou \textit{et al.} (2012), which has been modeled by a mass flow rate boundary condition in the simulations.}\]
assumed to satisfy Fourier’s and Fick’s laws, respectively. The ideal gas equation of state is used to close the system of equations.

Three different operating conditions are considered at a constant speedline of 64 krpm, see Tab. 1. Case D corresponds to a stable condition. For the other cases, the mass flow rate is restricted with purpose to provoke flow instabilities known as rotating stall and surge. The total pressure ratio TPR and the isentropic efficiency $\eta$ were previously evaluated Sundström et al. (2015) to assess the global performance of the centrifugal compressor stage. These parameters were found to be in good agreement with experimental data Guillou et al. (2012).

Upstream of the bell mouth inlet, the fluid is assumed to be approximately quiescent with known quantities of stagnation temperature and pressure. A non-reflective treatment of the inlet boundary is considered, see Giles (1988);
Table 1: Operating conditions with target mass flow rate and target static pressure at the outlet provided by experimental measurements.

<table>
<thead>
<tr>
<th>Case</th>
<th>$m$ (kg/s)</th>
<th>$p_{exit}$ (kPa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>0.050</td>
<td>165</td>
</tr>
<tr>
<td>B</td>
<td>0.085</td>
<td>165</td>
</tr>
<tr>
<td>D</td>
<td>0.280</td>
<td>156</td>
</tr>
</tbody>
</table>

Saxer (1992). The hemispherical inlet surface is located about three diffuser diameters upstream from the bell mouth entrance to the compressor as shown in Fig. 1. Adiabatic non-slip walls are considered on all solid surfaces.

The implicit Large Eddy Simulation (LES) approach is considered for resolving the large scale structures in the flow field. The governing equations are cast into continuous integral form and then formulated for discrete finite control volumes that constitute the computational domain. The transient term is discretized with a second-order temporal scheme. Convective flux and diffusive flux terms are discretized with a hybrid third order spatial scheme, which blends between upwind and central formulation.

Due to the geometrical complexity an unstructured polyhedral grid is considered consisting of approximately $9 \cdot 10^6$ cells. The near wall region is resolved at non-slip walls with prismatic cell layers due to the large wall normal velocity gradient. 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 flow-impeller interaction the domain is divided in a stationary region and a rotating region with a sliding interface in between Ferziger & Peric (2012). The two sides remain implicitly coupled and the cell connectivity is recalculated at each 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 grid of the non-stationary part is made to rotate at a constant angular velocity with respect to the compressor axis.

The Ffowcs Williams Hawkings (FW-H) analogy enables extrapolation of the sound emitted from a simulated flow scenario to the far-field. The idea is to estimate small pressure fluctuations at receiver locations. Different source estimation formulations of the FW-H models have been proposed suitable for different applications. In this study the Dunn Farassat Padula 1A formulation is used Farassat et al. (1987), which is customized for rotor blade applications.

The integrated emission surface is assumed to coincide with the impeller surface such that a monopole source evolves due to the thickness surface term and a dipole source arise from the blade loading surface term. The blade loading surface term $p'_L(\bar{x}, t)$ results from fluid displacement due to the impeller rotation. The thickness surface term $p'_T(\bar{x}, t)$ represents unsteady transport of the surface force distribution on the impeller. The quadrupole term $p'_Q(\bar{x}, t)$
accounts for turbulent mixing as a volumetric source, which is modeled by a "collapsing-sphere" formulation, as suggested by Brentner & Farassat (1998). For computational efficiency this is cast into the "source-time-dominant" algorithm from Casalino (2003). In this formulation the time derivatives are shifted inside the source term integrals, which circumvent numerical time differentiation of the integrals.

3. Results

Cross-correlations have been carried out between pressure signals acquired at different locations in the compressor. The purpose was to establish correlations between the observed modes inside the compressor (IS1 at the impeller’s inlet, D0 in the vaneless diffuse) and those dominant in the upstream hemispherical domain (Pin1 and Pin2 in the near-field acoustic region) or in the outlet pipe (Pout1), respectively. See also Fig. 1 for location of the monitoring points. The Cross Power Spectral Densities (CPSD) of the sampled pressure signal between the monitoring points are presented in Fig. 2, exhibiting the distinctive character for different operating conditions. The Rotating Order (RO) represents a non-dimensional frequency, and when it is equal unity it corresponds to the angular velocity of the impeller shaft. A tonality at RO = 10 including its higher harmonic is present for all operating conditions. It represents the blade passing frequency (BPF) since the impeller has 10 blades. The CPSD for Cases A and B reveal another tonality at RO = 0.5, which is not captured for the stable operating Case D. A peak at RO = 0.04 (43 Hz) is visible solely for Case A, with its higher harmonic being present only at some probe locations. Locations deep in the compressor (Fig. 2 (c) and (d)) exhibit a similar broadband CPSD with the aforementioned case-specific tonalities. Further upstream towards the compressor inlet region (Fig. 2 (a) and (b)), the differences between the amplitudes calculated for the unstable scenarios (A and B) and the stable Case D increase, in particular within the low and mid-frequency range. The largest amplitudes are found for Case A. One can note also that the spectra in Fig. 2(a) correspond to the near-field acoustic region, which is dominated by acoustic pressure fluctuations.

The CPSD spectra based on the pressure fluctuation data in the entrainment region, upstream of the compressor inlet is depicted in Fig. 2(a). A hump in the low-frequency range with peak amplitude at RO = 0.5 is visible. A second amplification establishes at a higher frequency at approximately RO = 3. This behavior has been observed for ducted fans configurations, see e.g. Shur et al. (2016). Moreover, the low frequency noise propagating in the upstream direction got amplified by changing the configuration of the duct inlet from a bell-mouth inlet to a rectangular one.

Figure 3 presents the instantaneous hydrodynamic and propagating acoustic pressure fluctuations for Case A at particular observed frequencies in the spectra. The acoustic waves traveling upstream form hemispherical waves with circumferential distributed highs and lows due to the interference effects. Pressure
fluctuations in the hydrodynamic region of the domain are also distinguishable. These are visible in the shear-layer developed at the bell-mouth inlet towards the compressor as well as in the impeller region and are associated with small scale turbulence related fluctuations. These have inherently higher magnitudes than acoustic waves.

Figure 4 illustrates the decay behavior of the RMS pressure fluctuations data as we are traveling upstream (away from the impeller - a hydrodynamic dominated region) towards the acoustic region. The data are extracted along a line aligned with the symmetry center-line of the impeller, starting from the impeller hub and going towards the inlet hemispherical boundary located in the acoustic region. The observed exponential decay is characteristic to the transition from hydrodynamic to acoustic regions and has been previously exposed for turbulent jets, see Suzuki & Colonius (2006).

The normalized pressure fluctuations (RMS) inside the compressor, shown for selected frequency-bands in Fig. 5, 6 and 7, indicate the locations of specific acoustic noise production and hence, their generation mechanism. For high frequencies $RO > 0.6$, the highest averaged pressure fluctuations can be seen to
Acoustic signature of flow instabilities in radial compressors

Figure 3: Hydrodynamic and acoustic pressure fluctuations obtained from the solution of the compressible governing flow equations for Case A, data sampled at specific frequencies (i.e. RO at 0.5, 3 and 10). Spherical waves at different wave lengths characteristic to the specific mentioned frequencies are manifested in the upstream far-field and the discharge outlet pipe. Hydrodynamic pressure fluctuations at larger magnitudes are observed inside the compressor. For the location of the post-processing planes see Fig. 1. Here, the P1 plane is positioned to the right of the front plane.

Figure 4: Root mean square of pressure fluctuation as function of distance from the impeller source along the center axis towards the acoustic near-field region.

occur in the proximity of the trailing blade tips for all operating conditions. High magnitudes occur due to flow mixing especially in the blade passages and the radial diffuser. The pressure fluctuations associated with the frequency band of RO = 0.3 – 0.6 distinguish the Cases A and B from Case D. A ring-like structure of high pressure fluctuation amplitudes can be observed in the cross-sectional plane located upstream of the impeller in the entrainment region for Cases A and B (visible in the inlet section and the P1 plane in Fig. 5, 6 and 7). These are
the signatures of flow vortical structures circulating upstream of the impeller, which have been described in a previous study Sundström et al. (2015). These flow features are linked with the presence of the rotating stall instability, also visible in the diffuser (at RO = 0.5). High amplitude fluctuations underneath the volute tongue can be noted in the band of RO = 0.3 − 0.6 for Case B, while these appear more amplified in a lower frequency band, i.e. RO = 0.1 − 0.3, for Case A. The band of RO = 0.1 − 0.3 exhibits further high pressure fluctuation amplitudes in the blade passages. Case A can be differentiated from the other cases by the low frequency pressure fluctuations i.e. RO = 0 − 0.1. The flow phenomenon surge occurring at a frequency of RO = 0.04 (43 Hz) is primarily responsible for the fluctuation amplitudes, which can be described as a cyclic filling and emptying process between the outlet plenum (or pipe) and the leading edge of the compressor. For a more detailed description of the flow phenomena see e.g. Sundström et al. (2014) or Sundström et al. (2015).

Figure 5: Normalized pressure fluctuation (root mean square - RMS) distribution for Case A around specified frequency bands. For the location of the post-processing planes see Fig. 1. Here, the cross-sectional P1 plane upstream of the impeller is presented to the right of the front plane.

The coherence of fluctuations (static pressure and velocity) is investigated via two point space-time cross correlation analysis. The rotating stall frequency (RO = 0.5) is clearly visible for the Cases A and B as shown in the space-time
correlation diagrams in Fig. 8, which are based on the pressure and axial velocity fluctuations, respectively.

The blade passing frequency can be noted in the pressure fluctuations for all cases. For Case D, the blade passing frequency is the most correlated feature. The slope \((z/t)\) of the peak pressure correlations (see Fig. 8, upper row) is the group speed for the instability propagation, which in this case is the speed of sound (340 m/s). Moreover these disturbances based on pressure fluctuations are propagating upstream, away from the impeller region. For the data based on axial velocity fluctuation for Cases A and B in Fig. 8, this characteristic is seen to be dominated by horizontal lines, which are identified as disturbances due to rotating stall. The space-time cross-correlation plots expose significant time-scales of dominant disturbances. Horizontal distributions can be interpreted such that disturbances appear at similar times.

A similar analysis based on cross-correlations of fluctuating pressure and tangential velocity component, respectively, carried out along a circumferential line inside of the diffuser (i.e. mid of the diffuser channel) is presented in Fig. 9. An interesting observation is that the momentum and pressure disturbance do not always exhibit the same time scales and hence are not always driven by the same disturbance. The data based on pressure for Cases B and D shows
several disturbances or characteristic lines, one for each fluctuation disturbance associated with the blade passing. Their slopes are slightly different as compared to the upstream propagating disturbances depicted in Fig. 8, resulting in a group speed slightly larger 380 m/s than the speed of sound at ambient conditions. Towards the off-design operating conditions, Cases A and B, the slope of the characteristics is seen to be modulated by low-frequency disturbances associates with rotating stall (note that the cross-correlations depicted in Fig. 9 are based on the circumferential velocity). Around the diffuser for Cases A and B, disturbances are seen, which correspond to crests and valleys associated with a convected rotating stall. Between the positions at 12 o’clock and under the volute tongue the rotating stall slope characteristic is dominant. Some variation is seen, which suggests that the convection speed of the rotating stall depends on the circumferential orientation. It is slightly speeding up in the region under the volute tongue and is slowing down after that. This effect may be caused by the local blockage due to the presence of the small recirculation flow region under the volute tongue. For Case A and the correlation based on pressure disturbances, the pattern shows horizontal lines. This is identified as a standing wave and thus associated with surge operation (note the different time length considered for capturing the flow frequency of surge in Case A).
Figure 8: Space-time diagrams; normalized two point cross correlation along an upstream line. Top row based on pressure and bottom row based on axial velocity fluctuations, respectively. The sketch at the bottom clarifies start and end positions of the upstream line used for the cross correlation computation. The mid location along the line is used as a reference point for the correlation. The color range is from -1 to 1.

A two point space-time cross correlation based on pressure disturbances is also presented along a line in the volute outlet pipe, see Fig. 10. For Cases D and B, clear characteristics due to the blade passing are seen, with group speed close to the sound speed. A standing wave is shown for Case A, associated with surge.

In order to separate the sources leading to acoustic noise generation from the hydrodynamic pressure fluctuations, the FW-H approach is employed. The sources considered with the FW-H acoustic formulation are the blade loading (dipole) term, the thickness (monopole) term, and the quadrupole term.
Figure 9: Normalized cross correlation along a circumferential line in the diffuser at $r = 60$ mm. Top row based on pressure and bottom row based on tangential velocity fluctuations, respectively. The start and end points of the circumferential line as well as the reference point, are at $\theta = 0$, i.e. at 12 o’clock, see sketched inset for clarification purposes. The color range is from -1 to 1.

However, the applicability of the FW-H approach in confined configurations is doubtful, c.f. Åbom (2006); Farassat et al. (1987). Therefore, the results obtained by using the FW-H approach are compared against those obtained directly from the compressible LES calculations (previously validated in Sundström et al. (2014, 2015)). The comparisons are in terms of Overall Sound Pressure Levels OASPL [dB] and SPL spectra obtained at the upstream located probe point Pin2. Table 2 shows that the maximal difference between the directly calculated OASPLs and those based on the FW-H approach is of 3 dB. Figure 11 shows the spectra comparisons for the three different operating conditions using the direct approach and the FW-H acoustic analogy. A fair
Acoustic signature of flow instabilities in radial compressors

Figure 10: Normalized cross correlation based on pressure fluctuation along a line in the outlet pipe. The reference location is at \( x = 0 \) mm. The color range is from -1 to 1.

<table>
<thead>
<tr>
<th>Case</th>
<th>( SPL_L )</th>
<th>( SPL_T )</th>
<th>( SPL_Q )</th>
<th>( OASPL_{FW-H} )</th>
<th>( OASPL_{direct} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>132 dB</td>
<td>24 dB</td>
<td>78 dB</td>
<td>132 dB</td>
<td>130 dB</td>
</tr>
<tr>
<td>B</td>
<td>130 dB</td>
<td>25 dB</td>
<td>73 dB</td>
<td>130 dB</td>
<td>127 dB</td>
</tr>
<tr>
<td>D</td>
<td>120 dB</td>
<td>37 dB</td>
<td>60 dB</td>
<td>120 dB</td>
<td>117 dB</td>
</tr>
</tbody>
</table>

Table 2: Overall sound pressure level (OASPL) in (dB) at probe point Pin2 computed with FW-H and direct approach. Acoustic source contribution from blade loading \( p'_L \), thickness \( p'_T \) and quadrupole \( p'_Q \) terms are depicted for Case A, B and D. The overall sound pressure level (OASPL) with the FW-H is computed using: 

\[
OASPL_{FW-H} = 10\log_{10} \left( 10^{\frac{SPL_T}{10}} + 10^{\frac{SPL_Q}{10}} + 10^{\frac{SPL_L}{10}} \right) \text{dB}.
\]

agreement (within 10 dB) is found at RO frequencies below four. However, the agreement is not as good at the highest frequencies. One has to keep in mind that with the FW-H approach the radiated sound from the acoustic sources is propagating towards the acoustic far-field without being affected by the surrounding geometry of the compressor.

Table 2 clarifies the relative importance of the individual source terms of the FW-H approach. Note that the acoustic sources due to blade loading and blade thickness are existing on the surface of the blades, while the quadrupole acoustic source exists in the volume of the flow.
Their spatial distributions in the vicinity of the impeller are illustrated in Fig. 12. The blade loading term, $p'_L$, is induced by the impeller interacting with flow structures and represents the essential source dominating the sound pressure level contribution by an order of magnitude (see Tab. 2). Especially at the leading and trailing edges, where the impeller crops up unsteady flow structures, the blade loading term reveals high amplitudes. The thickness surface term, $p'_T$, establishes, in contrast to the blade loading term, towards the trailing edge of the blade suction sides where the flow separates. This term has the least important contribution to the overall sound pressure levels as shown in Tab. 2. The quadrupole term, $p'_Q$, represents turbulent fluctuations leading to acoustic noise propagation, which is mainly generated due to separated flow from flow mixing downstream of the blades.

The effect of large-scale flow structures on the blade loading term can be analyzed by filtering the flow field and blade loading term at a particular frequencies associated with the observed instabilities. Figure 13 shows the phase angle evolution, $\Phi$, of the reconstructed acoustic blade loading term (top row) for Case A at the low surge frequency tonality (RO = 0.04). In this figure, correspondent flow data are included (bottom row). It shows how the coherent flow structure in surge affect the blade loading term, which in turn affects the momentum flow into the radial diffuser for one complete surge cycle. At $\Phi = 3\pi/2$ is subjected to the filling phase of surge and streamlines illustrates downstream flow with a moderate incidence flow angle. The corresponding blade loading term depicts a neutral level. At $\Phi = 0$, the streamlines depict growing incidence flow angle, which causes a building up of the blade loading level from the mid to the rear end of the impeller blades. By $\Phi = \pi/2$ the compressor is seen to swap towards the emptying phase, and the pressure thus drops. The effect on the flow field is that the flow reverses mid-way in the blade passages depicting large separation zones. From the phase when the blade loading is at
its minimum the flow field gradually recovers with more downstream directed flow.

Figure 14 shows the phase angle evolution, $\Phi$, of the flow perturbation (without mean flow) and the blade loading term contribution at $RO = 0.5$. In the P1 plane upstream of the impeller, circulating vortical structures can be observed, which have been identified as rotating stall. These vortical structures are cropped by the blades causing high amplitude contributions to the blade loading term at the leading edge. This can be observed by the high magnitude noise sources shown in Fig. 14 for all phase angles. Depending on the directionality of the vortex rotation, the flow incidence angles at the blades and hence, their loading is influenced. This causes corresponding unsteady pressures at the back wall of the impeller.

At zero phase angle, $\Phi = 0$, two high-pressure zones are located at 12 and 6 o’clock near the leading edges. These correspond to the co-rotating vortex pair in the P1 plane with streamlines depicting outwards spiraling. In between, low-pressure perturbation zones can be found and they correspond to the other co-rotating vortex pair with inward spiraling. A similar distribution with two high pressure zones and two low pressure zones in between can also be seen towards the blade trailing edges at zero phase angle. A low pressure region remains between four and six o’clock at other phase angles, while the other propagate around the circumference. This asymmetry can explain the subharmonic tonality at $RO = 0.25$. This asymmetry is related to the asymmetric time averaged pressure field at off-design operating conditions (see e.g. Sundström et al. (2014)). In the region between 4 and 6 o’clock, the highest flow momentum is discharged into the diffuser (Sundström et al. 2014),
which acts as a sink for the low-pressure region. The corresponding flow field of the rotating stall can be found downstream in the diffuser and the volute regions. By a phase angle $\Phi$ of $\pi/2$ further in the cycle, the impeller surface is rotated clockwise one quarter of a revolution. However, the pressure distribution of the rotating stall feature rotate half as much, i.e. $\pi/4$, but also clockwise. In other words, in one impeller revolution, the rotating stall feature moves clockwise only half a revolution. Thus, this spinning motion circumferentially in the absolute frame explains the tonality found at $RO = 0.5$ in the CPSD spectra. A high pressure zone corresponds to streamlines depicting radial outgoing diffuser flow. A low pressure zone relates to reversing inward directed flow. By $\Phi = \pi$ the two high pressure zones close to the inducer are now horizontally oriented and by $\Phi = 3\pi/2$ they are oriented diagonally between 10 and 4 o’clock. From $\Phi = 3\pi/2$ until $\Phi = 2\pi$ the pressure and velocity distribution will gradually come back to the description stated for $\Phi = 0$, and the rotating stall cycle will repeat.

![Phase evolution of acoustic blade loading term (top row) and the modal flow perturbation (bottom row) at the surge frequency RO = 0.04 for Case A. The mean velocity is not superimposed. A phase angle $\Phi = 0$ represents the start of the surge cycle whereas $\Phi = \pi/2$ indicates half way of a surge cycle.](image-url)
At the rotating order, i.e. the angular velocity of the impeller shaft, a distinct characteristic dipole distribution is observed for all investigated cases. This is presented in Fig. 15 only for Case D and $\Phi = 0$. The characteristic is similar with the $RO = 0.5$ mode described above, but exposing a distinct regularity. This means that the $RO = 1$ mode shape describes a circumferential rotation relative the impeller surface. The fluctuation level is particularly focused towards the exducer as well at the leading edges and the blade tips. This mode is seen to have several higher harmonics, where the $RO$ equal 2, 3, and 4 are also presented. The dipole surface source distribution at $RO = 1$ including its higher harmonics are seen to correlate with the tonalities observed in the SPL spectra, shown in Fig. 11.
4. Conclusions

Aerodynamic and aero-acoustic properties in a vaneless diffuser centrifugal compressor were investigated for one constant speed-line by gradually restricting the mass flow rate going from near optimum design efficiency towards off-design surge operating conditions. The compressible flow was computed using the LES approach assuming a full annulus and grid motion of the impeller region. Assessment of flow acoustic coupling was performed using Fourier surface spectra and two-point cross-correlation analysis. The FW-H equation was used for classification of acoustic sources.

Computations of the unsteady flow in a centrifugal compressor at low mass flow rates revealed connections between particular flow instabilities and their acoustic signature. The mass flow rate ingested by the compressor was gradually reduced to provoke the instabilities, i.e. rotating stall and surge.

Distinct tonalities in the SPL spectra were correlated with acoustic source terms of the FW-H analogy. The blade loading source term, with dipole characteristic, was found to be several orders of magnitudes larger as compared to thickness noise and quadrupole source terms. In fact, most noise is found to be generated at the impeller due to non-uniform blade loading. The computed acoustic power scaling does support the theoretical scaling law via Lighthill’s acoustic analogy. In other words, for subsonic flow dipole sound power is stronger than the quadrupole since the scaling indicates $W_d : W_q \sim 1 : M^2$. Moreover, monopole may only be important close to sonic tip speeds, as shown experimentally by Raitor & Neise (2008).

For distinct tonalities found in the SPL spectra, the spatial pattern of the blade loading term on the impeller surface were investigated using modal decomposition methods. For the second most restricted mass flow rate considered (Case B), rotating stall instability was provoked. The SPL spectra include an additional tonality at RO = 0.5. In the high-frequency range (RO > 2), the SPL spectra calculated for the unstable flow mass flow scenarios were found to be similar with the SPL spectrum at the stable operating condition. The RO = 0.5
Acoustic signature of flow instabilities in radial compressors

Mode shape exposes a characteristic dipole distribution. The entrained vortical flow structures are chopped up at the impeller leading edge. The perturbed flow angles provoke a non-uniform pressure loading on the blades, where the high momentum flow between four and six o’clock acts as a low pressure sink. At surge operating conditions this sink becomes amplified and only one low pressure perturbation runs around the circumference at the discharge. Therefore, fluctuations of RO = 0.5 and RO = 0.25 (underneath the volute tongue) are related.

Low frequency instability at a fraction of the rotating order RO = 0.04 describing a filling and emptying process was captured at the most restricted mass flow rate case considered (Case A). This is in agreement with published literature on the subject where the mode is referred to a system instability named surge, see e.g. Semlitsch & Mihăescu (2016); Semlitsch et al. (2013); Galindo et al. (2015). The SPL spectra expose a distinct tonality at this frequency and it correlates with a coherent blade loading feature distributed on the impeller surface. The acoustic sources are distributed symmetrically in a circumferential arrangement with focus towards the rear part of the impeller surface as well as towards the leading edges and blade tips.

In the mid frequency range, a second amplified broadband intensity is observed in the SPL spectra between RO = 2 and four times the angular velocity of the impeller. This is in agreement with Tomita et al. (2013); Evans & Ward (2005) who discuss this feature and designate it as “whoosh noise”. However, the LES data only expose a camelback shape in the far field spectra. Such feature cannot be seen deeper inside the compressor. Therefore, it is not necessarily connected with “whoose noise”. In Torregrosa et al. (2014) it is mentioned that broadband noise occurs in the range 1 < RO < 2 for a similar size compressor as the one used in the present study. Whereas Raitor & Neise (2008) relates this to “tip clearance” noise due to leakage between the shroud and the blade tips. Towards low mass flow rate operating conditions, the time-averaged pressure field becomes more asymmetric. This causes the broadband acoustic noise levels to increase by about 2-4 dB. This manifested in static perturbation modes at all frequencies. The SPL point spectra shows narrowband features in this frequency interval, i.e. RO > 1. They corresponds to the tonality RO =1 and higher harmonics to RO = 1. This correlates with a number of harmonic modes shapes found with modal decomposition at relevant magnitudes. The RO = 2 mode is seen to be significant in this frequency interval (1 < RO < 4).

Acknowledgments

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 is greatly appreciated. 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. The authors are grateful to Dr. Gutmark and his team at University of Cincinnati for sharing the experimental data as well as the geometry.
REFERENCES

ÅBOM, M. 2006 An introduction to flow acoustics. TRITA-AVE 2006:04, ISSN 1651-7660, ISRN/KTH/AVE/N-06/04-SE.


GILES, M. 1988 Non-reflecting boundary conditions for the Euler equations. Computational Fluid Dynamics Laboratory, Department of Aeronautics and Astronautics, Massachusetts Institute of Technology.


Paper 5
Evaluation of Centrifugal Compressor Performance Models using Large Eddy Simulation Data

Elias Sundström¹, Bertrand Kerres² and Mihai Mihăescu¹

¹ The Competence Center for Gas Exchange (CCGEx), KTH Mechanics, S-100 44 Stockholm, Sweden
² KTH Machine Design, S-100 44 Stockholm, Sweden

ASME Technical paper, GT2016-57169 (2016)

Zero-dimensional (0D) compressor performance models, which consist of several sub-models for different loss terms, are useful tools in early design stages. In this paper, one typical model for centrifugal compressors is evaluated by comparing the loss-terms predicted by the model to data extracted from experimentally validated Large-Eddy-Simulation. The simulations were run on a truck-sized turbocharger compressor with a ported shroud and a vaneless diffuser. Four operating points are considered: One mass flow at design conditions and one mass flow close to surge, on two speedlines. The performance prediction models evaluated are impeller incidence loss, impeller skin friction loss, diffuser skin friction loss, and the tip clearance loss. Results show that the total losses are well-predicted by the model at design conditions. Friction losses are approximately independent of mass flow in the LES data, while the 0D model assumes a quadratic increase. The assumption of constant tip clearance loss is validated by the LES data, and the impeller incidence loss model also fits the data well. Due to the ported shroud, most of the losses as calculated by entropy increase occur through iso- baric mixing at the impeller inlet.
**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\omega$</td>
<td>Angular frequency (rad/s).</td>
</tr>
<tr>
<td>$r$</td>
<td>Radius (m).</td>
</tr>
<tr>
<td>$U$</td>
<td>Tangential velocity component (m/s).</td>
</tr>
<tr>
<td>$C$</td>
<td>Absolute velocity (m/s).</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>Absolute flow angle (deg).</td>
</tr>
<tr>
<td>$\beta$</td>
<td>Relative flow angle (deg).</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>Mass flow rate (kg/s).</td>
</tr>
<tr>
<td>$h$</td>
<td>Enthalpy (J/kg).</td>
</tr>
<tr>
<td>$s$</td>
<td>Entropy (J/kg-K).</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Efficiency.</td>
</tr>
<tr>
<td>$Re$</td>
<td>Reynolds number.</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>Ratio of specific heat.</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Air density (kg/m$^3$).</td>
</tr>
<tr>
<td>$p$</td>
<td>Pressure (Pa).</td>
</tr>
<tr>
<td>$T$</td>
<td>Temperature (K).</td>
</tr>
<tr>
<td>$R$</td>
<td>Gas constant (J/kg-K).</td>
</tr>
<tr>
<td>$c_p$</td>
<td>Specific heat at constant pressure (J/kg-K).</td>
</tr>
<tr>
<td>$n, \theta, m$</td>
<td>Meridional coordinates.</td>
</tr>
</tbody>
</table>

**Subscript**

- $i$: Impeller or incidence.
- $d$: Diffuser.
- $0, t$: Total or stagnation value.
- $1$: Impeller inlet.
- $2$: Impeller outlet.
- $s$: Isentropic state change.
- $b$: Blade
- $c$: Compressor.
- $f$: Friction.

**INTRODUCTION**

The European goal for vehicle emission standards is a 50% CO$_2$ reduction by 2020. This puts the automotive industry under pressure to improve the fuel economy of internal combustion engines (ICE). One possibility is engine downsizing and turbocharging, which improves the energetic efficiency and reduces emissions. Reduced engine speed also improves fuel economy but demands that the turbocharger operates closer to off-design condition at low mass flow rates. This limits the compressor performance and introduces unwanted flow oscillations, e.g. surge, see experimental study by Guillou *et al.* (2012) and numerical assessment by Sundström *et al.* (2014). The surge condition induces large pressure pulsations in the compressor system and may even cause flow reversal through the impeller. This may lead to vibrations with high stress levels on blades and casing as well as connected piping. Operation under surge condition is therefore unwanted since it may damage the compressor.
Overall the unsteady surge condition results in a highly complex unsteady flow field, where several flow-driven instabilities develop, e.g. adverse pressure gradients, shear-layer instabilities, wake effects and boundary layer separation. These phenomena are difficult to quantify, and exemplifies why the surge inception mechanism is not yet fully understood. To prevent the centrifugal compressor operating in unstable surge condition a physics based understanding of the triggering mechanism of both rotating stall and surge is needed for a successful surge suppression control design. High quality flow visualization is possible by means of experimental Particle Imaging Visualization (PIV) technique but is currently restricted to areas with optical access due to the confined space. Time resolved measurements with mounted pressure transducers offers point spectral analysis to distinguish narrowband and broadband flow features.

Numerical simulation of the Navier-Stokes equations with broadband enabled turbulence, e.g. Large Eddy Simulation (LES), allows access to high quality flow visualization everywhere inside the compressor system. When thoroughly validated it offers access to a physics based understanding of the surge inception mechanism in areas out of reach when experimental methods are considered. However, there are numerous modeling challenges with this approach, e.g. high Reynolds number flow ($10^5 < Re < 10^7$), complex geometry, complex physics, rotating stall calls for $360^\circ$ with grid rotation, large range of temporal length scales (surge at $\mathcal{O}(1)$ Hz compared with blade passing frequency at $\mathcal{O}(10^4)$ Hz) demands long runs times on multiprocessor supercomputers. There are sources of numerical errors with grid and schemes used, e.g. need for turbulence closure, and adopting boundary condition to reflect the experimental set-up. Assessment of uncertainties is a multidisciplinary effort required between Computational Fluid Dynamics (CFD) and measurements, e.g. large number grid points required for grid independence results. LES produces large amount of data which introduce post-processing challenges. Therefore, LES is not a fast affordable method for early design of centrifugal compressors, and motivates development of zero-dimensional loss models that provides reasonable prediction of compressor characteristics as part of a larger ICE system. Validity assessment of the loss models is challenging due to the limitation of obtaining experimental data for internal compressor flow.

This paper analyze the flow loss prediction model as presented by Gravdahl & Egeland (1999), which is based on energy conservation through the compressor stage. Individual loss terms are included such as the incidence loss term from works by Futral & Wasserbauer (1965), friction loss term according to Ferguson (1963), ported shroud mixing loss term from works by Greitzer et al. (2007), and finally blade loading loss term according to Thanapandi & Prasad (1990). These loss terms account for some of the entropy increase in the compressor stage. To verify the applicability of the loss prediction models, validated Large Eddy Simulation (LES) data is presented for a centrifugal compressor composed with a ported shroud. The motivation is to identify regimes in the compressor map.
where the model deviates from LES data and study the flow structures in the LES flow field data resulting in unaccounted entropy increases, not considered adequately with the included flow loss prediction terms.

**COMPRESSOR SYSTEM**

Figure 1 shows the centrifugal compressor system under consideration and Tab. 1 lists some design parameters. Measured angles are relative to the tangent and the blade inlet angle $\beta_{1b}$ is the angle at the root mean square inducer radius $r_1 = \sqrt{(r_{h1}^2 + r_{t1}^2)}$.

The impeller has ten main blades and no splitter blades. The main blades are designed with backsweep at the radial exit. Air at ambient condition enters the compressor via a bell mouth inlet and is directed to the impeller eye. At the radial exit the cross-section area increases and the air is being gradually compressed through an annular diffuser through a diffusion process and the gas is decelerated with the resulting increase in the static pressure. A volute is attached at the diffuser exit which serves to collect the diffuser exit flow and guide it to the volute exit pipe outlet. A ported shroud is fitted at the stationary upstream pipe casing and is supported by four asymmetrically arranged ribs. In case of flow reversal in the impeller at off-design condition (e.g. surge) some of this flow between the impeller tip and shroud is ventilated through the open ports and is guided into the ported shroud cavity. There it is guided further upstream and is mixed and recirculated with delivered gas upstream from the bell mouth inlet. This is a so called passive surge control device with the purpose to lower the peak pressure rise and damp pressure oscillation and thereby widening the operation range towards the surge line in the compressor map.

**COMPRESSIBLE FLUID FLOW AND HEAT TRANSFER**

The mass, momentum and energy conservation equations, respectively, are considered and solved with a finite volume approach using the CFD code STAR-CCM+ developed by CD-adapco. The governing equations for the compressible flow results in a non-linear system with more unknowns than equations. It is closed using the ideal gas law, which provides a link between the momentum equations and the energy equation. 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. The viscosity is obtained from Sutherland's equation.

**Large Eddy Simulation**

The three-dimensional, time-dependent large-scale turbulent motion responsible for the fluid mixing in the compressor flow, is 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. The filtered convection term is modeled by introducing the sub-grid scale (SGS) stresses. 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. 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 (discretized with a second-order temporal scheme), as well as the convective flux, the diffusive flux and the source terms, respectively (discretized with a bounded third-order spatial scheme). The role of the SGS model is to dissipate enough kinetic energy at smallest sub-grid scale through a turbulent energy cascade. Since this effect is obtained from the truncation error an implicit LES with no explicit SGS is considered and hence no explicit filter is required. The validity of the implicit LES approach can be evaluated by verifying the power spectral density of the velocity fluctuations in a point exhibiting isotropic turbulence and analyzing the characteristic -5/3 energy decay slope in the inertial subrange, see previous works in Ref. Sundström et al. (2014) and Sundström et al. (2015).
Table 1: PARAMETERS OF THE CENTRIFUGAL COMPRESSOR.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller</td>
<td></td>
<td></td>
</tr>
<tr>
<td>No. of blades (Z)</td>
<td>10</td>
<td>-</td>
</tr>
<tr>
<td>Inlet radius tip (r_{t1})</td>
<td>33</td>
<td>mm</td>
</tr>
<tr>
<td>Inlet radius hub (r_{h1})</td>
<td>9</td>
<td>mm</td>
</tr>
<tr>
<td>Exit radius (r_2)</td>
<td>43</td>
<td>mm</td>
</tr>
<tr>
<td>Exit width (b_2)</td>
<td>4.6</td>
<td>mm</td>
</tr>
<tr>
<td>Blade inlet angle (\beta_1)</td>
<td>30</td>
<td>deg</td>
</tr>
<tr>
<td>Blade exit angle (\beta_2)</td>
<td>68</td>
<td>deg</td>
</tr>
<tr>
<td>Tip clearance (r_c)</td>
<td>1</td>
<td>mm</td>
</tr>
<tr>
<td>Blade tip width (b_w)</td>
<td>0.5</td>
<td>mm</td>
</tr>
<tr>
<td>Diffuser</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Outlet radius (r_3)</td>
<td>76</td>
<td>mm</td>
</tr>
<tr>
<td>System</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Exit pipe volume (V_p)</td>
<td>0.0018</td>
<td>m^3</td>
</tr>
<tr>
<td>Duct area (A_1)</td>
<td>0.0018</td>
<td>m^2</td>
</tr>
<tr>
<td>Duct length (L_c)</td>
<td>930</td>
<td>mm</td>
</tr>
</tbody>
</table>

**Boundary Conditions**

Upstream of the bell mouth inlet the fluid is assumed to be approximately quiescent with known quantities of the stagnation temperature and pressure, respectively. However, the impeller rotation introduce hydrodynamic turbulent fluctuations at significant energy levels which manifest in both broadband and narrowband features in the spectra. For a correct treatment the inlet boundary is therefore considered non-reflective for incoming sound waves. In the experimental setup the outlet mass flow rate is regulated with a throttling valve. Both temperature and pressure as assumed to float at this station but the mass flow rate is approximately steady. Since, the geometry of the throttling valve were not available as part of the this study a fixed mass flow is applied at the outlet and temperature and pressure are extrapolated from upstream characteristics when the flow is purely outgoing. As conditions approach 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 is employed. This helps suppress potential backflow and stabilizes the numerical approach, yet flow reversal in the vicinity of the volute exit cone is allowed with this treatment. 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. Adiabatic non-slip walls are used in the compressor. Velocity components at the walls as well as the temperature gradient from the wall to the boundary layer are hence zero.
Numerical Grid

The computation domain is discretized in finite control volumes. Due to the geometrical complexity (e.g. high surface curvature at leading impeller blades as well as thin gaps and tight corners) an unstructured polyhedral mesh consisting of approximately $9 \cdot 10^6$ cells is considered. The near wall region is resolved at non-slip walls 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 separates from the impeller surface and therefore such boundaries are resolved with no use of wall functions. Thereby a more accurate prediction of the separation point is 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. 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.

Zero-Dimension Model

The zero-dimension loss prediction model considered is based on the works by Gravdahl & Egeland (1999). This model is thus based on energy conservation through the compressor stage. Figure 2 shows two velocity triangles, one for the incoming fluid entering the inducer and one for the fluid leaving at the impeller exit and entering the diffuser. These are sketched with respect to a curvilinear coordinate system attached to a meridional curved surface that cuts through the rotating impeller region and follows the hub and the shroud curvature. Since the flow is guided by the hub and the shroud it is therefore assumed, in the model derivation that follows, that the normal velocity component to this surface is small and negligible. This is an assumption that is approximately valid for a steady-state flow. This simplification means that flow velocity is only considered in coordinates $m$ and $\theta$. At the radial distance $r_1$ the tangential velocity at the inducer is calculated as:

$$U_1 = r_1 \omega \quad (1)$$

The absolute velocity at the inducer $C_1$ is given by:

$$C_1 = \frac{\dot{m}}{\rho_1 A_1} \quad (2)$$

The fluid is guided by the impeller blade and leaves at the impeller tip (i.e. at radius $r_2$) at the absolute velocity $C_2$ and the tangential tip velocity is $U_2$. 

Figure 2: VELOCITY TRIANGLE AT INDUCER AND EXDUCER.

Ideal Energy Transfer

The impeller rotation results in a change of angular momentum of the fluid which is given by:

\[ \tau_c = \dot{m}(r_2 C_{\theta 2} - r_1 C_{\theta 1}) \]  

where \( \tau_c \) is the compressor torque and \( C_{\theta 2} \) is the tangential component of the absolute gas velocity \( C_2 \). The applied torque delivers power to the fluid as:

\[ \dot{W}_c = \omega \tau_c = \omega \dot{m}(r_2 C_{\theta 2} - r_1 C_{\theta 1}) = \dot{m}(U_2 C_{\theta 2} - U_1 C_{\theta 1}) = \dot{m}\Delta h_{0,\text{ideal}} \]  

where \( \Delta h_{0,\text{ideal}} \) is the ideal specific enthalpy change in the fluid excluding losses.

For an impeller without backsweep, the tangential velocity \( C_{\theta 2} \) leaving the exducer should equal \( U_2 \). For impellers with backsweep, the outlet tangential velocity is obtained by

\[ C_{\theta 2,\text{id}} = U_2(1 - \phi_2 \cot \beta_{2b}) \]  

where \( \phi_2 = C_{m2}/U_2 \) is the tip flow coefficient. This ideal velocity is reduced due to slip, where the flow is deflected away from the direction of the impeller rotation. The slip factor is given as follows:

\[ \sigma = C_{\theta 2}/C_{\theta 2,\text{id}} \]  

In the works by Stanitz & Ellis (1950) the slip factor for a radial impeller is approximated as \( \sigma \approx 1 - 2/Z \). In reality, the slip factor can depend on
more parameters than the number of blades (e.g. passage geometry, inducer to exducer radius ratio and mass flow rate etc). The specific work input from (4), using Eq. (6) is thus:

\[ \Delta h_{0c,ideal} = \frac{\dot{W}_c}{\dot{m}} = \sigma U_2^2 (1 - \phi_2 \cot \beta_{2b}) \]  

### Energy Losses in the Compressor

From Eq. (7) it can be observed that \( \Delta h_{0c,ideal} \) is a function of speed and mass flow rate. Moreover, the enthalpy rise is not constant due to various losses. From Watson & Janota (1982) these arise mainly from incidence loss in the impeller \( \Delta h_{ii} \) and friction losses in impeller and diffuser, \( \Delta h_{fi} \) and \( \Delta h_{fd} \). These losses are essential in defining the stability characteristics of the compressor. Some other losses such as tip clearance loss and loss due to back flow will be taken into account in the computation of the compressor efficiency.

#### Incidence Loss.

From Watson & Janota (1982) the incidence loss is approximated as:

\[ \Delta h_{ii} = \frac{1}{2} U_1 - \frac{\cot \beta_{1i} \dot{m}}{\rho_0 A_1} \right)^2 \]  

The assumption for this is that the incoming relative velocity at the inducer \( W_1 \) instantaneously change its direction to align with the blade angle (i.e. from \( \beta_i \) to \( \beta_{1b} \)) and a part of the kinetic energy from the tangential component \( W_{\theta 1} \) is lost according to:

\[ \Delta h_{ii} = \frac{(U - W_{\theta 1})^2}{2} \]  

#### Friction Loss.

According to Ferguson (1963), the impeller friction loss can be calculated as follows:

\[ \Delta h_{fi} = f_i \frac{l_i}{D_i} \frac{W_{1b}^2}{2} \]  

where \( f_i \) is the friction factor, \( l_i \) is the mean channel length and the mean hydraulic channel diameter is computed as \( D_i = 2\pi r_1/Z \). Haaland’s explicit formula (Haaland 1983) is used for calculation of the friction factor:

\[ 1/f_i^{1/2} = -1.8 \log(6.9/Re + [(\epsilon/D_i)/3.7]^{1.1}) \]  

where the Reynolds number is computed as \( Re = U_2 b_2/\nu \), with \( b_2 \) as the impeller tip width and \( \nu \) as the kinematic viscosity. From Fig. 2 it is seen that:

\[ \frac{W_{1b}}{\sin \beta_1} = \frac{W_1}{\sin \beta_{1b}} \]  

Combining Eqns. (12), (2) and (10) gives:

\[ \Delta h_{fi} = \frac{f_i l_i}{2D_i \rho_1^2 A_1^2 \sin \beta_{1b}} \dot{m} = k_{fi} \dot{m}^2 \]
Thus, the friction loss varies like the square of the mass flow rate and is independent of the impeller speed $U$. The diffuser friction loss can be modeled in a similar fashion as:

$$\Delta h_{fd} = k_{fd} \dot{m}^2$$

(14)

From the quadratic variation of the friction loss is can be seen that it has a stabilizing effect of the compressor characteristics since a higher friction loss will shift the surge line to the left, but on the expense of a lower pressure rise.

**Efficiency.**

The isentropic efficiency of the compressor stage can be computed as:

$$\eta_i(\dot{m}, U_1) = \frac{\Delta h_{0c,ideal}}{\Delta h_{0c,ideal} + \Delta h_{loss}}$$

(15)

where

$$\Delta h_{loss} = \Delta h_{ii} + \Delta h_{ijf} + \Delta h_{df}$$

(16)

Combining the various losses and adding losses associated with clearance and backflow losses the corrected isentropic efficiency is as follows:

$$\eta_i(\dot{m}, U_1) = \frac{\Delta h_{0c,ideal}}{\Delta h_{0c,ideal} + \Delta h_{loss}} - \Delta \eta_{b,f} - \Delta \eta_c - \Delta \eta_d$$

(17)

The clearance loss from Pampreen (1973) is approximated as $\Delta \eta_c = 0.3(r_c/b)$, where $r_c$ is the radial clearance and $b$ is the impeller tip width. The backflow loss $\Delta \eta_{b,f} = 0.03$ is according to Watson & Janota (1982), which does not included effects from ported shroud mixing loss. According to Cumpsty (1993) the volute loss $\Delta \eta_v \approx 0.03$ is due to the radial kinetic energy lost between the diffuser and the volute. The efficiency drop due to incomplete diffusion in the diffuser $\Delta \eta_d \approx 0.03$ is according to Cumpsty (1993). All the efficiency loss terms are constants and hence independent of the mass flow rate.

**Energy Transfer and Pressure Rise**

The pressure rise from the inducer to the exducer $p_{01} \rightarrow p_{02}$ can be described as first an isentropic process (e.g. $\Delta h_{0c,ideal} = \Delta h_{0c} - \Delta h_{01}$), followed by an isenthalpic (or adiabatic) process with entropy increase $\Delta s > 0$. The entropy increase can be computed from the enthalpy losses in Eqns. (16, 17) by assuming an isobaric process from point $h_{02}$ to $h_{02s}$ and integrating $ds = dh/T$. Thus, the total energy transfer is given by:

$$\Delta h_{0c}(U_1, \dot{m}) = \Delta h_{0c,ideal} - \Delta h_{loss}$$

(18)

where $\Delta h_{0c}$ has a maxima at some mass flow rate corresponding to optimum efficiency and drops for higher and lower mass flows, see Fig. 3. Under the assumption of a calorically perfect gas and an isentropic process together with the energy equation it can be shown that the stagnation pressure ratio is given by:

$$\frac{p_{02}}{p_{01}} = \left(1 + \frac{\eta_i \Delta h_{0c,ideal}}{T_{01} cp} \right)^{\gamma/(\gamma-1)}$$

(19)
where the losses are included and $p_{02}/p_{01}$ is the overall total pressure ratio characteristic of the compressor.

**COMPARISON WITH VALIDATED LES DATA**

Figure 4 shows the compressor characteristics computed with the 0D model and compared with LES and experimental data for two speedlines ($N = 64000$ and 88000 rpm). The cases considered for comparison are shown in Tab. 2. The experimental data is according to Guillou *et al.* (2012). The vertical bar in the LES data indicates the time variation in monitored pressure ratio and efficiency, respectively. The horizontal bar represents the variation in monitored mass flow rate.

The surge line in Fig. 4 is defined as the line connecting the local maxima of the constant speedlines. Naturally, there are many uncertainties in the 0D model loss coefficients as well as the many assumptions adopted which leads to large deviations as compared to experimental data and LES (e.g. almost
Table 2: OPERATING CONDITIONS.

<table>
<thead>
<tr>
<th>Case</th>
<th>$N$ (rpm)</th>
<th>$\dot{m}$ (kg/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>S64 (Surge)</td>
<td>64000</td>
<td>0.05</td>
</tr>
<tr>
<td>NS64 (Near-surge)</td>
<td>64000</td>
<td>0.085</td>
</tr>
<tr>
<td>D64 (Design)</td>
<td>64000</td>
<td>0.28</td>
</tr>
<tr>
<td>NS88 (Near-surge)</td>
<td>88000</td>
<td>0.166</td>
</tr>
<tr>
<td>D88 (Design)</td>
<td>88000</td>
<td>0.28</td>
</tr>
</tbody>
</table>

Figure 4: CENTRIFUGAL COMPRESSOR CHARACTERISTICS COMPUTED WITH 0D MODEL AND COMPARED WITH LES AND EXPERIMENTAL DATA.

...no modeling assumptions). Nevertheless, some of the major characteristics has been captured with the 0D model and the next step is to scrutinize how loss terms compare with the LES data.
Evaluation of Centrifugal Compressor Performance Models

Computation of overall enthalpy losses

The mass flow average total enthalpy and entropy is computed from upstream of the impeller to a station at the exit of the diffuser for design and near-surge operation at two different speedlines, see Fig. 5. The ports were excluded in the computation of these averages. All cases show an approximate isobaric process upstream of the blade leading edge. The extend of the isobaric entropy increase here depends on the impeller speed as well as on the mass flow. Comparing two cases on the same speedline in Fig. 5, e.g. cases NS64 and D64, one can see that the compressor mass flow has a large influence, with lower mass flow leading to a higher entropy increase. For a constant mass flow, a higher impeller speed results in higher entropy increase at the inlet, as observed from comparing cases D64 and D88. After the mixing at the inlet, the entropy also increases over the impeller due to flow losses in the design case D64. In the near-surge case NS64, on the other hand, the impeller flow seems almost isentropic, with a reduction in entropy in the inducer. A similar observation can be made comparing cases D88 and NS88. From the exducer edge up to the diffuser exit the process is approximately isenthalpic with a rise in entropy. From Fig. 5 it can be noted that the impeller leading edge is located at the transition from the isobaric curve to the isenthalpic increase of \( h_0 \). Moreover, the impeller trailing edge is located at the transition with increase in entropy only. Cases NS64, D88, and NS88 show a drop in enthalpy at the impeller outlet, and some enthalpy increase in the diffuser. Since the simulation was run with adiabatic walls, this effect must be due to the averaging procedure used. The isobaric curves in Fig. 5 are obtained from considering the entropy change \( ds \) in the fluid:

\[
 ds = \frac{dh}{T} - R\frac{dp}{p} 
\]

In the isobaric case \( dp = 0 \). Integrating Eq. (20) yields:

\[
 T_0 = T_{0ref} \cdot e^{\Delta s/c_p} \Rightarrow h_0 = h_{0ref} \cdot e^{\Delta s/c_p} 
\]

which relates the total enthalpy with the change in entropy. The total enthalpy loss \( \Delta h_{loss} = \Delta h_{02s} - \Delta h_{02} \) is given from the difference between the curves \( p_{02s} \) and \( p_{02} \).

Effect of the ported shroud

The ported shroud will be investigated in the following part. Their effect is a reversal of high-entropy flow through the ports. This shifts a major part of the entropy increase in the impeller to the mixing of the inflow with the port flow. The different stations for computing the entropy rise are shown in Fig. 6. Station 0 is the incoming mass flow, which is mixed with the backflow through the port as taken at station \( p \). The mixing takes place between stations 1 and 1m, respectively.

Figure 7 gives a schematic illustration of the effect of the mixing of the inlet and port mass flow, and also illustrates a reference impeller without a ported shroud. Starting at the impeller inlet, the backflow from the ports with a high
specific entropy mixes approximately isobarically with the incoming flow from the atmosphere. The impeller then performs work on the fluid, adding enthalpy and entropy. At station 1e, a part of the fluid leaves the impeller through the ports. Since the fluid leaving has a higher-than-average entropy, the mean
entropy decreases, which leads also to a small increase in total pressure. After the ports, the flow through the impeller continues along a similar path in the h-s diagram as for a normal impeller without ports. The effect of the ports on the h-s path of the flow is thus to shift some of the losses that occur in the impeller upstream.

These effects can be observed in Fig. 5 as well. The ratio of port mass flow to inlet mass flow is listed in column 2 of Tab. 3. A positive value means that the flow in the ports travels upstream. The mass flow through the ports is relatively smaller in the design mass flow cases D64 and D88. Only a part of the impeller losses is therefore accounted for at the mixing of port mass flow and inlet flow. In the near surge cases, on the other hand, where much more high-entropy fluid flows backwards through the ports, almost the complete impeller losses are accounted for through the inlet mixing, and the impeller itself seems to work isentropic. In case NS88, the reduction of the mean entropy in the impeller at station 1e can clearly be seen in Fig. 5. The same phenomenon is also visible for cases D88 and NS64, but less clear here.

The entropy rise due to the ported shroud mixing loss can be modeled using Eq. (22), cf. e.g. Greitzer et al. (2007), which gives a non-dimensional entropy increase through the mixing of two streams. The mixing losses occur between station 1 and station 1m, see Fig. 6. While mixing could take place further

Figure 7: SCHEMATIC OF ISOBARIC MIXING OF INLET AND PORT MASS FLOW.
downstream as well, station 1m needed to be located upstream the impeller in order to better distinguish between mixing effects and impeller effects. The results are listed in Tab. 3. For cases NS64 and D88, the modeled entropy increase due to this mixing explains ca. 90% of the whole entropy increase upstream the impeller, while it explain ca. 70% of the losses in cases D64 and NS88. For case D64, this could an artifact of the low absolute values, so that the relative error from averaging becomes large. For case NS88, on the other hand, there must be some additional losses that are not accounted for using Eq. (22). This could for example be due to backflow along the shroud, which is not included in the port mass flow and entropy.

\[
\frac{T_1 \Delta s_{mix}}{C_0^2} = \left[ \frac{1}{(\gamma - 1) M_0^2} \right] \left[ \frac{\dot{m}_0}{\dot{m}_1} \ln \left( \frac{T_{1m}}{T_{0}} \right) + \frac{\dot{m}_p}{\dot{m}_1} \ln \left( \frac{T_{1m}}{T_{tp}} \right) \right] + \frac{p_{t1} - p_{t1m}}{\rho_0 C_0^2} \tag{22}
\]

The model in Eq. (22) seem to be able to capture the losses generated by the port backflow, provided a good estimate of port mass flow and thermodynamic state is available.

<table>
<thead>
<tr>
<th></th>
<th>(\dot{m}_p/\dot{m}_0)</th>
<th>(s_p) (J/kg-K) (Eq. 22)</th>
<th>(\Delta s_{mix}) (J/kg-K)</th>
<th>LES (Fig. 5)</th>
</tr>
</thead>
<tbody>
<tr>
<td>NS64</td>
<td>0.54</td>
<td>2590</td>
<td>88</td>
<td>97</td>
</tr>
<tr>
<td>D64</td>
<td>0.08</td>
<td>2460</td>
<td>9</td>
<td>13</td>
</tr>
<tr>
<td>NS88</td>
<td>0.31</td>
<td>2690</td>
<td>115</td>
<td>165</td>
</tr>
<tr>
<td>D88</td>
<td>0.23</td>
<td>2540</td>
<td>51</td>
<td>55</td>
</tr>
</tbody>
</table>

**Comparison of individual loss terms**

In the following the losses from friction and tip clearance are compared with LES data. The enthalpy loss due to friction in the LES data is computed as follows:

\[
\Delta h_f = \frac{1}{\dot{m}} \int_S \tau_w \cdot C_w dA \tag{23}
\]

where \(\tau_w\) – wall shear stress and \(C_w\) represents the cell relative velocity near the wall. The integration is performed on every surface area element for both the impeller and the diffuser parts. It should be noted that \(C_W\) is not an integral velocity in the boundary layer and is dependent on the near-wall grid resolution. So the chosen form in Eq. (23) is only an approximation of the enthalpy loss due to friction.

The incidence loss is computed as the change in total enthalpy due to the instantaneous change of the incidence flow angle for a corresponding comparison
with Eq. (8). In the works by Moussa (1978) and Aungier (1995) the tip clearance loss is estimated as follows:

$$\Delta h_{pc} = \frac{2\dot{m}_{cl}\Delta p_{cl}}{\dot{m}_1}$$

(24)

where $\dot{m}_{cl}$ is the tip clearance mass flow rate and $\Delta p_{cl}$ is the pressure difference across the gap. Figure 8 compares enthalpy losses from the LES data and and the predicted data from the 0D model. The computed incidence loss is

Figure 8: ENTHALPY LOSSES COMPUTED WITH LES DATA AND THE 0D MODEL (REPORTED UNITS IN J/kg×10^4).

approximately in the same order of magnitude between the LES data and the 0D model and shows a consistent trend with increasing magnitude towards off-design conditions. However, one complication is that the incidence loss depends on the blade angle which is a function of the radial direction. This effect is not included in the 0D model. Moreover, the impeller and diffuser friction losses are of similar magnitude at near-surge as well as at design mass flow condition. The friction loss in the 0D model varies with the square of the
mass flow rate and is approximately valid for attached pipe flow. In contrast, the LES data shows only a marginal variation with respect to the mass flow rate. Separation reduces the near-wall velocity, which implies less skin friction loss. It can also be seen that the combined losses from incidence and friction is approximately one order of magnitude smaller than the total loss $\Delta h_{\text{loss}}$ which suggest that dominant losses are due to the residual loss terms $\Delta h_{\text{res}}$ (i.e. the additional correcting loss terms in Eq. (17)). The total enthalpy loss increases in the LES data as well in the 0D model towards off-design conditions. This trend is correctly captured, and implies that it is less efficient to operate at near-surge as compared to design condition.

CONCLUSIONS

A zero dimensional model was evaluated for computing the characteristics of a centrifugal compressor with ported-shroud and vaneless diffuser. The efficiency and pressure rise in the compressor stage was based on energy transfer. Individual enthalpy loss terms from incidence, friction and other losses was compared with validated LES data for near-surge and design operating conditions for two different speedlines. The total enthalpy and the change in entropy were computed with the LES data from a station upstream of the impeller to the exit of the diffuser. For all considered operating conditions the LES data shows an initial approximate isobaric process, which is followed by an approximate isentropic process which accounts for most of the total enthalpy increase in the compressor stage. From the exducer edge up to the diffuser exit the process is approximately isenthalpic with a rise in entropy, a consequence from assumption of adiabatic walls. The effect of the ported shroud can be seen as a reversal of the high entropy flow through the ports, which shifts all entropy increase in the impeller to the isobaric mixing of the inflow with the port flow. Incidence and friction losses are seen to be in the same order to magnitude between the LES data and the 0D model. The LES data shows only a marginal friction variation with respect to the mass flow rate whereas the 0D model includes a friction variation like the the square of the mass flow rate. Reliable computation of friction losses calls for an integral velocity in the boundary layer. In the present computation, the friction loss is approximated using the cell relative velocity. Nevertheless, the combined losses from incidence and friction is approximately one order of magnitude smaller than the total losses in the compressor stage. The most dominant losses are accounted by other losses such as tip clearance loss and backflow loss. Despite an error from computing the friction with LES the effect on the total enthalpy is small. Finally, the LES data shows an increased total enthalpy towards off-design conditions which implies that it is more efficient to operate at design condition as compared to off-design condition. A similar trend is seen with the 0D model.

Acknowledgment

This work was conducted at the Competence Center for Gas Exchange (CCGEx). The authors would like to thank the Swedish Energy Agency (STEM), Volvo
Cars, Volvo GTT, and SCANIA for financial support. The Swedish National Infrastructure for Computing (SNIC) via HPC2N and PDC, the Parallel Computing Center at KTH, are acknowledged for providing the computing resources. A special thanks to Dr. Gutmark and his team at University of Cincinnati for sharing experimental data as well as geometrical data of the centrifugal compressor.

REFERENCES


Ferguson, T. 1963 *The centrifugal compressor stage*. Butterworths.


Paper 6
Analysis of vaneless diffuser stall instability in a centrifugal compressor

Elias Sundström\textsuperscript{a} and Mihai Mihăescu\textsuperscript{a} and Marco Giachi\textsuperscript{b} and Elisabetta Belardini\textsuperscript{b} and Vittorio Michelassi\textsuperscript{b}

\textsuperscript{a} The Competence Center for Gas Exchange (CCGEx), KTH Mechanics, S-100 44 Stockholm, Sweden
\textsuperscript{b} GE, Oil&Gas, Florence, Italy


Numerical simulations based on the Large Eddy Simulation approach were conducted with the aim to explore vaneless diffuser rotating stall instability in a centrifugal compressor. The effect of the impeller blade passage was included as an inlet boundary condition with sufficiently low flow angle relative to the tangent to provoke the instability and cause circulation in the diffuser core flow. Flow quantities, velocity and pressure, were extracted to accumulate statistics for calculating mean velocity and mean Reynolds stresses in the wall-to-wall direction. The paper focuses on the assessment of the complex response of the system to the velocity perturbations imposed, the resulting pressure gradient and flow curvature effects.

Nomenclature

$C_p$ pressure coefficient $\frac{2}{\gamma M_{ref}^2} \left( \frac{P}{P_{ref}} - 1 \right)$

$P_{ref}$ reference pressure at the inlet [Pa]

$M_{ref}$ reference Mach number at the inlet [-]

$r, \theta, z$ cylindrical coordinates defined in Fig. 1

$m, n$ meridional and wall normal coordinates defined in Fig. 1

$U_r, U_\theta, U_m$ mean radial, tangential and meridional velocities [m/s]

$U_{tip}$ blade tip velocity [m/s]

$u'_r, u'_\theta, u'_m$ radial, tangential and meridional fluctuating velocities [m/s]

$f_{IMP}, f_{BPF}$ impeller angular velocity and blade passing frequency [Hz]

$N$ number of rotating flow cells or number of grid points

$b$ diffuser channel width at the inlet radius $R_2$ [m]

$R_2$ diffuser inlet radius [m]

$\chi$ blade-to-blade stagger angle [radian]

1. Introduction

Ideally, for an efficient design the centrifugal compressor provides a steady fluid flow at an elevated pressure with minimal losses. However, the performance of
the centrifugal compressor is harmed under the off-design operating conditions occurring at low flow rates. In such circumstances flow instabilities are developed through the compressor, which may cause large fluctuations in the flow variables and even introduce vibrations with high structural stress levels into the machine. This is an unwanted situation and the aim is therefore to understand the physical mechanisms responsible for the developed flow instabilities. Through linear stability analysis, see e.g. Jansen (1964), the diffuser rotating stall is said to occur when the local return flow angle relative the meridional exceeds $90^\circ$. However, a diffuser instability generally develops earlier than $90^\circ$, i.e. prior to purely tangential flow. There exist some correlations that indicate the instability angle to be around $75^\circ - 85^\circ$ w.r.t. the radial direction depending on the operating conditions, see e.g. Abdelhamid (1980). The theory Jansen (1964) also states that it occurs with steady, uniform inflow and outflow conditions and thus independent of the interaction with the rotating impeller. A criticism is that inflow periodic unsteadiness may trigger a diffuser instability, see e.g. Madhavan & Wright (1985). Another criticism is that the theory assumes diffuser flow as inviscid. This implies that the dynamic character of diffuser rotating stall depends on the inviscid core flow and is independent of viscous effects, dominating in the boundary layer. It is important to acknowledge some observations where the periodic unsteadiness may be delaying stall, see e.g. Seifert et al. (1996) where low pressure turbine suction side boundary layer stall, and axial compressor stall were investigated.

For the actual evolution of the dynamic character and thus prediction of the number of rotating stall cells and their propagation speed around the annulus, different research groups have considered a time-evolving calculation assuming 2D-unsteady, inviscid and incompressible flow, see e.g. Tsujimoto et al. (1996); Ljevar et al. (2006); Kalinkevych & Shcherbakov (2013). The effect of the blades on the flow is fed at the diffuser inlet boundary and a Dirichlet condition with static pressure is considered on the outlet surface. One observation is that the evolution of the rotating stall cells depends on the specified outlet boundary condition; see e.g. Dawes (1995). Specification of a Dirichlet boundary condition at the outlet was seen to result in strong gradients of the velocity field. This is in contrast to the linear stability analysis where the Neumann boundary condition is used, which assumes zero pressure gradients in both the radial and tangential directions. In scenarios with reversing flow at the outlet boundary special consideration is needed for an appropriate specification of the backflow direction. This could be extrapolated or it could be treated as normal to the face elements on the outlet surface. In the ideal case, the outlet boundary would be extended further downstream to prevent recirculation. In the numerical work of Tsujimoto et al. (1996), extrapolating the backflow direction was seen to be in agreement with experimental results of Wachter & Rieder (1985) for a wide vaneless diffuser with uniform inflow. The rotating stall feature was illustrated as two high static pressure cells at low velocity with reversing flow and two lower pressure cells at higher outgoing velocity. These cells were reported to be rotating at sub-synchronous rotational speed. In a real diffuser design the
channel width is narrow where viscous effects in the boundary layer cannot be neglected. This is due to the growth of a displacement thickness that may be different on the wall boundaries, depending on the quality of the flow delivered by the impeller flow, see e.g. Guidotti et al. (2013). Additionally, the rotation of the impeller with high blade tip speed results in a high Reynolds number flow that enters the diffuser. Therefore the flow is thus turbulent with large fluctuations in the flow quantities, and subsequently with high shear rate, which may become unstable. Typically, the fluctuations are amplified at off-design conditions, as documented by Gaetani et al. (2012). These effects are not taken into account with the 2D inviscid approach. In the work by Dazin et al. (2008) the Navier-Stokes equations were averaged with the Reynolds decomposition resulting in the need for turbulence closure modeling. They employed a two equation isotropic turbulence model. The numerical result was found in good agreement with observed pressure fluctuations and number of rotating stall cells as compared with Particle Imaging Velocimetry (PIV), see Dazin et al. (2011). However, it was concluded that the numerical result using isotropic turbulence modeling underestimates the strength of the rotating stall vortex as compared with PIV measurements. This may be explained by excessive turbulent diffusion, a known issue of linear turbulence models in presence of adverse pressure gradients, and issues associated with predicting accurately features such as boundary layer separation. Many different closure models exist for calculating turbulent flow. In the case of the diffuser flow there is evidently some argument for including curvature effects in the modeling approach, since curvature requires discerning turbulence anisotropy. Consequently, it is clear that the turbulence model should take anisotropic effects into consideration on top of considering an approach able to resolve the large, energy containing flow structures. A computational efficient approach, to include streamline-curvature effects in the framework of eddy-viscosity turbulence modelling, is introduction of a correction factor to the turbulent production term in the turbulent kinetic energy equation, see e.g. Shur et al. (2000). This factor depends on the strain-rate as well as the rotation-rate tensor including several proposed modelling constants. Due to the large freedom in choosing the modelling constants a small variation to proposed default values may potentially yield significant differences in the predicted turbulence production. Nevertheless, eddy-viscosity modelling with curvature correction showed good predictive quality compared to experimental data for a vaneless diffuser application during rotating stall conditions, as shown in Marconcini et al. (2017).

Previously it was demonstrated that the angular momentum instability associated with curvature plays a key role in provoking large effects on the turbulent boundary layer, see work by Clausen et al. (1993). The boundary layer is affected by the radial pressure gradient, radial curvature near the inlet and tangential straining as the flow cross section area increases. Since the flow is swirling it is therefore needed to include 3D effects and the destabilizing tangential curvature.
The present paper aims to capture the evolution of the rotating stall cells and their propagation around the annulus by employing the Large Eddy Simulation (LES) based approach to a 3D annular vaneless diffuser configuration with a turnaround outlet. The possibility of capturing anisotropic turbulence effect in the boundary layer will help elucidate the variation of velocities and Reynolds stresses in the swirling diffuser flow. Especially in key areas susceptible close to separation and recirculation in order to investigate the effect on fluctuation levels. For a statistical assessment the flow field is quantified by means of flow mode decomposition techniques. A sensitivity study to the outlet boundary conditions imposed will be carried out. The numerical methodology is described in the next section, which is followed by presentation of results. The relevance of the obtained results to the scientific and designer community is included in a concluding summary section.

2. Numerical methodology

The narrow annular vaneless diffuser with a turnaround outlet, shown in Fig. 1, contains a converging section close to the impeller blades. The overall extent before the turnaround in the radial direction is ≈ 1.5\(R_2\). The curvature of the turnaround is approximately 0.2\(R_2\) and the outlet is extruded to a location at 0.82\(R_2\). The effect of the impeller blade passing is provided as meridional and tangential velocity profiles, respectively. The 3D time-dependent large-scale turbulent motion responsible for the fluid mixing in the vaneless diffuser is computed using the Large Eddy Simulation (LES) approach. The computational domain is discretized with a curvilinear grid which constitutes the elements of the finite volume methodology employed. Via Taylor series expansion an approximate representation is obtained for the transient, convective and diffusive terms respectively. The temporal term is discretized with a 2nd order scheme, which uses the solution from the current solution and from the previous two time levels. A hybrid 2nd order upwind/central differencing scheme is used for the convective term. The diffusive flux term is treated with a second-order expression, which involves the cell center value, the neighboring cell center value, and including the diffusion flux at the interior face. The introduced dissipative truncation error is known to mimic Smagorinsky-type SGS modeling, see works by Margolin & Rider (2002); Fureby & Grinstein (2002). This is a rational for using an implicit LES with no explicit SGS. The validity of an implicit LES approach can be verified by evaluating the energy decay slope for a point exhibiting isotropic turbulence in the inertial subrange, i.e. \(-5/3\) for a velocity fluctuating component. However, such condition is not expected for a narrow channel flow under pressure gradient and strong flow curvature yielding significant effect on the boundary layer development and with associated separation. Therefore, code validation is assessed alternatively by means of a grid dependency study, which is presented further below.

A constant-density turbulent flow is assumed with a Reynolds number \(Re_b \approx 1e5\) and inlet reference Mach number \(M_{ref} = 0.2\). The reference flow angle at the inlet boundary is relative to the tangential. It is computed in a
point between the blades and at the diffuser channel midpoint between the shroud and hub side:

\[ \alpha_{IN} = \arctan \left( \frac{U_m}{U_\theta} \right) \] (1)

At the inlet boundary \( U_m \) and \( U_\theta \) have variable distributions in both axial and tangential directions, with purpose to mimic a real velocity distribution from an upstream impeller. Figure 2 shows the meridional and tangential velocity profile distributions scaled with \( U_{tip} \) for two different reference inlet flow angle cases \( \alpha_{IN} = 9.5^\circ \) and \( \alpha_{IN} = 5.5^\circ \). For each blade tip width segment circumferentially around the inlet diffuser plane, \( U_\theta = U_{tip} \) and \( U_m = 0 \). This has the effect of modeling the blade blockage effect, which is presented as tangential distributions for two blade passages, see last row in Fig. 2 for orientation purposes. In addition, the velocity distribution is made to rotate at the impeller angular velocity \( f_{IMP} \). Comparing peak values in Fig. 2 it can be seen that the amplitude of the flow distortion entering the domain is in the order of 0.1\( U_{tip} \) at the distinct blade passing frequency \( f_{BPF} \). Additional stochastic fluctuation is not considered, due to the overwhelming effect of the periodic unsteady incoming wakes described in Fig. 2. At the outlet boundary, the pressure is fixed when the flow is outgoing. In case of backflow the pressure and flow direction is extrapolated from the
outlet boundary. At the inlet the velocity components are specified and pressure is allowed to float. Therefore, the pressure is fixed in the scenario with pure outflow at the pressure outlet cell faces.

For validity of the numerical result a grid dependency study is performed on three different grids named; coarse, medium and fine (see also Fig. 3). A grid refinement factor of two is used between grids. The wall resolution for the fine grid considered is: $\Delta n = 0.5$, $\Delta m = 30$, and $\Delta \theta = 30$. The normalized wall distances are computed as the surface average of the hub and shroud walls, respectively, between meridional stations $m = 0$ and $m = 1$. For a fair comparison the time-step size is adjusted between grids so that the time
averaged convective Courant number is unity. Approximately 30 through flow times were simulated for each case. Five through flow times were needed to establish a suitable initial turbulent flow field. Therefore, data corresponding to 25 through flow times are used for statistical analysis of the flow field, which is adequate to capture a low frequency instability in the order of $0.02f_{BPF}$. The integrated pressure rise $\Delta C_p$ is computed as the difference in area averaged pressure coefficient at meridional stations $m = 0$ and $m = 0.8$, respectively. This parameter is evaluated with respect to the grid resolution in the meridional direction $\Delta m^+$. Figure 3 indicates that the solution (i.e. $\Delta C_p$) approach an asymptotic limit value as the grid is refined. The Richardson 2nd order extrapolation shows that the fine grid resolution is marginally different compared to a hypothetical infinite grid resolution. It is also evident that the relative error w.r.t. the infinite grid solution drops monotonically two orders of magnitude for each refinement order. This result correlates with the chosen second order scheme and an expected truncation error of $O(\Delta m^{+2})$.

![Graph showing $\Delta C_p$ vs $\Delta m^+$](image1)

**Figure 3:** Grid dependency assessment based the on $\Delta C_p$ between $m = 0$ and $m = 0.8$ for three different spatial resolutions. The Richardson 2nd order extrapolation indicates the hypothetical infinite grid solution. The relative error is seen to reduce two orders magnitude w.r.t. to the grid size $\Delta m^+$.

### 3. LES data analysis

Based on the grid dependency assessment, the solution obtained with the fine grid resolution is chosen for further analysis. Figure 4 shows distributions on the side view of the time-averaged pressure, velocity components, and Reynolds stress levels, respectively, for the flow angle $\alpha_{IN} = 9.5^\circ$. The Reynolds stresses are scaled as a fluctuation intensity w.r.t. the mean local velocity. This velocity is obtained at the midway distance between the shroud and the hub and along

<table>
<thead>
<tr>
<th></th>
<th>$\Delta n^+$</th>
<th>$\Delta m^+$</th>
<th>$\Delta \theta^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarse</td>
<td>10</td>
<td>120</td>
<td>120</td>
</tr>
<tr>
<td>medium</td>
<td>5</td>
<td>60</td>
<td>60</td>
</tr>
<tr>
<td>fine</td>
<td>0.5</td>
<td>30</td>
<td>30</td>
</tr>
</tbody>
</table>
the meridional coordinate. A characteristic pressure increase with a relatively steep gradient is seen until the top of the turnaround. A pressure recovery follows towards the outlet, which is due to the reduced flow cross section. At the turnaround a small pressure gradient is present in the wall-to-wall normal direction with slightly lower pressure on the hub side compared to the shroud side. The time-averaged meridional and tangential velocity profiles, respectively,

$$C_p, \frac{U_n}{U_m}, \frac{U_\theta}{U_0}, \sqrt{\frac{u_m^\prime}{U_m}}, \sqrt{\frac{u_\theta^\prime}{U_0}}, \sqrt{\frac{u_m^\prime u_\theta^\prime}{U_0}}$$

Figure 4: Time-averaged scalar contour distributions for $\alpha_{IN} = 9.5^\circ$, colored on the side view. Flow quantities of interest are indicated in the text annotations.

show a strong shear layer close to the inlet, which is due to the jet like shape of the imposed inlet boundary profile. In the straight section the velocity components gradually approach the shape of a fully turbulent developed flow profile with high fluctuation levels close to the walls. To help see this more clearly the time-averaged meridional and tangential velocity profiles, respectively, are plotted in Fig. 5 for different stations along the meridional direction, between $m = 0.05$ and $m = 0.8$. At meridional $m = 0.6$, just prior the turnaround, the profiles approach an approximate self-similar solution. Beyond $m = 0.6$ both profiles are seen to tilt with reduced through flow close to the hub wall, an effect caused by the turnaround bend. Note that the time-averaged meridional velocity has a more extreme slope and indicates tendency of an inflection point between $m = 0.7$ and $m = 0.8$. This leads to increased fluctuations (root mean square) of this component ($u_m'$. The steepness close to the shroud and hub at $m = 0.6$ is compatible with the law of the wall. Therefore, wall modeling with lower resolution near the wall may lead to acceptable predictions. However, in the vicinity of the developing inflection point of the meridional velocity component there is evidence that wall modeling would not be acceptable and compromise prediction of potential boundary layer separation.
Figure 5: Profiles of the time-averaged meridional and tangential velocity components for flow angle $\alpha_{IN} = 9.5^\circ$ for different locations along the meridional indicated in the figure. Profiles of mean meridional and tangential velocity fluctuation intensity are shown in the bottom row. Profiles are scaled with the mean local velocity.

The fluctuation components of velocity in the meridional, the tangential as well as the co-variant direction, respectively, are depicted in Fig. 4. A more detailed distribution of the fluctuation intensities is also depicted in Fig. 5 for different meridional stations. Close to the inlet, peak fluctuation intensity is located close to the wall. In the middle of the channel, the peak values reduce and approach self-similarity. One explanation for this is the shape of the imposed boundary condition with the blades passing by creating high stress levels in the wake flow. Further downstream however, the intensity of the fluctuation reduces, which indicates turbulence decay. It is noted that the Reynolds stress anisotropy component exhibits a sudden growth at $m = 0.7$ close to the hub wall (see Fig. 4). However, this follows by a rapid recovery when the flow enters the area of favorable pressure gradient towards the outlet. In general the tangential stress level is dominant but the meridional and anisotropic stress components are not insignificant. This clarifies the importance of employing a numerical approach that can capture anisotropic features in diffuser flow.

Assessment of the level of curvature requires a determination of the flow angle with respect to the tangent. This is presented in Fig. 6 at different meridional stations. The wall is said to be in a collateral state if the flow angle $\alpha = \arctan(U_m/U_\theta)$ is independent of the wall normal direction $n$. In other
words, there would be a linear relationship between tangential and meridional velocities. It appears that this may hold in the straight diffuser section. A collateral state does not seem to hold past the turnaround at $m = 0.7$. The figure shows evidence of a change in the flow angle as moving downstream with two different flow angles at the hub and shroud walls, respectively. In other words a smaller flow angle holds on the hub side as compared to the shroud side, and presenting a moderate gradient in between. This could be due to the concerted action of the turnaround bending effect and viscous effect near the wall, affecting the swirling motion of the core flow differently in the near wall regions associated with the hub and the shroud, respectively. In addition to this, the flow angle is seen to increase gradually towards the turnaround, which therefore gives a stabilizing effect. In the area of the turnaround the flow angle decreases close to the hub side, which is associated with reduced through flow. This clearly leads to a destabilizing effect of the flow.

The presented results so far have only exposed statistics of the flow variables. A further assessment is carried out analyzing the power spectral density (PSD) of a monitored flow quantity of interest. Figure 7a) shows PSD for the tangential velocity at $m = 0.1$ in the mid of the diffuser channel, i.e. $n = 0.5$. A combination of broadband and narrowband content is observed in the spectra. The tonality of the blade passing frequency including several higher harmonics is correctly captured with the numerical approach. In the low frequency end of the spectra for both inlet reference flow angles considered significant tonalities are revealed. For the larger flow angle case $\alpha_{IN} = 9.5^\circ$, one notable peak is observed at $0.04f_{BPF}$. For the lower flow angle case $\alpha_{IN} = 5.5^\circ$ two significant low frequency tonalities can be observed, one at $0.03f_{BPF}$ and the other at $0.07f_{BPF}$. The low frequency tonalities (i.e. $0.04f_{BPF}$ for the large flow angle case and $0.03f_{BPF}$ and $0.07f_{BPF}$ for the smaller flow angle case), correspond to flow modes with a periodic wave like character. For the lower flow angle case the intensity level in the low to middle frequency range is seen to amplify.
Moreover, the peak of the dominant narrowband feature, i.e. 0.07f_{BPF} for α_{IN} = 5.5° as compared to 0.04f_{BPF} for α_{IN} = 9.5°, is shifted to a higher frequency relative to the shaft speed. It should be noted that the sample length corresponds to approximately 25 cycles of the low frequency tonality. Therefore, these features appear as broad peaks rather than sharp peaks in the point spectra. However, this could be improved with a more generous sampling length and hence simulation elapsed time. Although, if a harmonic signal is Gaussian shaped, the variation is not exactly sinusoidal.

Figure 7: a) Power Spectral Density of the tangential velocity component in a probe point located at m = 0.1 in the mid of the diffuser channel n = 0.5. Data is compared for the two different flow angles considered. b) Cross Power Spectral Density of the tangential velocity component between probe points located at m = 0.1 and m = 0.7 in the mid of the diffuser channel n = 0.5. Data is compared for the two flow angles cases considered.

Up to this point the numerical data elucidate two possible mechanisms for onset of the low frequency instability, i.e. one shear layer close to the inlet and a tendency of an inflection point in the turnaround section. To correlate these mechanisms with the rotating instability the point-to-point cross power spectral density (CPSD) of the tangential velocity fluctuation is presented in Fig. 7b). The signals in the two different points show strong correlation at the rotating instability (i.e. 0.04f_{BPF} for α_{IN} = 9.5°) as well as at the blade passing frequency. The phase shift at the rotating instability frequency is approximately −χ. This means that the signal closer to inlet is ahead approximately one blade-to-blade passage. For the smaller flow angle case α_{IN} = 5.5° spatial coherency is also found in the low frequency range. There, the peak at 0.07f_{BPF} is seen to be more dominant over the peak at 0.03f_{BPF}. Since their frequency ratio is not a whole number integer they are not harmonics of each other.

For a statistical characterization of the flow mode associated with the interesting narrowband instability a Fourier surface spectra analysis is performed
of the velocity fluctuations $u'_r$ and $u'_\theta$ for every point distributed on a meridional surface located midway between the hub and the shroud, which follows the diffuser channel curvature. The number of cycles of the low frequency mode shape included in the surface spectra computation is limited to four and the result is presented in Fig. 8 for the two different flow angles considered. For a

\[
\alpha_{IN} = 9.5^\circ, 0.04f_{BPF} \quad \alpha_{IN} = 5.5^\circ, 0.03f_{BPF} \quad \alpha_{IN} = 5.5^\circ, 0.07f_{BPF} \quad \alpha_{IN} = 9.5^\circ, f_{BPF}
\]

\[
\frac{u'_r}{U_{tip}} \quad \frac{u'_\theta}{U_{tip}}
\]

-2 (%) 2 -3 (%) 3 -4 (%) 4 -2 (%) 2

Figure 8: Fourier surface spectra flow mode decomposition. Narrow-band radial (top row) and tangential (bottom row) velocity fluctuation intensity colored on a midway-section between hub and shroud which follows the diffuser curvature. In detail, this correspond to $n = 0.5, \forall m \in [0, 0.8], \forall \theta \in [0, 2\pi]$. In the figure this is presented in a frontal view, which means that points for meridional positions at the top of the domain and beyond towards the outlet are not seen. Rotating stall modes at different flow angles, $\alpha_{IN} = 9.5^\circ$ (first column) and $\alpha_{IN} = 5.5^\circ$ (second and third columns). Last column shows the blade passing frequency flow mode for $\alpha_{IN} = 9.5^\circ$.

physical interpretation the intensities of the flow modes should be observed as a superposition on the mean flow ($U_i$) where the subscript $i$ is a direction index. The real part of the flow mode spectra for the larger flow angle reveals two areas with negative intensity and two areas with positive intensity respectively, and distributed on opposing sides of each other. A positive radial intensity level is interpreted as outgoing flow and a negative level is a local inversion of the radial velocity component. The imaginary part (not included in the figure) is a phase shifted 90° representation of the real part in the rotation direction of the shaft (clockwise in the figure). With the real and imaginary parts added together $u'_i = \Re\{u'_i\} \cos(2\pi ft) - \Im\{u'_i\} \sin(2\pi ft)$ the shape of the mode is seen to rotate
in the same direction as the impeller rotation. In other words, the flow mode
can be described as a rotating instability consisting of two counter-rotating flow
cells relative to each other. This qualitative description is in agreement with
results of other research groups, e.g. Wachter & Rieder (1985); Tsujimoto et al.
(1996). The same feature can be attributed also to the lower flow angle (second
and third columns) but with a larger number of rotating cells distributed evenly
around the circumferential. The tonality at $0.03f_{BPF}$ for $\alpha_{IN} = 5.5^\circ$ (see
Fig. 7a)) shows three disturbances (second column in Fig. 8), whereas the other
peak at $0.07f_{BPF}$ shows five rotating cells (third column in Fig. 8).

One notable difference with the larger flow angle $\alpha_{IN} = 9.5^\circ$ as compared
to the lower flow angle $\alpha_{IN} = 5.5^\circ$, can be seen in terms of their most dominant
low frequency tonalities, respectively. That is $0.04f_{BPF}$ (first column in Fig. 8)
as compared to $f_{0.07}f_{BPF}$ (second column in Fig. 8). For the coherent mode
shape at $0.04f_{BPF}$ the intensity of the tangential fluctuating wave is more
dominant compared to the radial wave, which is rather weak in intensity. For
peak $0.04f_{BPF}$ in the lower flow angle case, this is reversed, i.e. the radial wave
dominates over the tangential. The reason to this can be attributed to type
and location of the outlet boundary condition for flow case $\alpha_{IN} = 5.5^\circ$. For
this case a fraction of the cell face elements on the outlet were subjected to
reversed flow, which means that those face elements can no longer be considered
as a real outlet boundary. Under such situation the numerical solution may not
be unique. Therefore, for the lower flow angle case there is a fraction of the outlet
that violates the well-posedness criteria, see Hadamard (1902). Strictly
speaking, a solution to a problem that is not well-posed does not make sense,
and so the numerical result for the lower flow angle case cannot be fully trusted.
The remedy to this is to consider a repositioning of the outlet where the flow is
purely outgoing. However, a common result is that the intensity level of the
rotating disturbance reaches a peak at the top of the turnaround and with a
growth point location prior to the turnaround.

The propagation speed of the rotating stall cells can be analyzed using
a space-time cross correlation. For this the tangential velocity fluctuating
component $u_\theta'$ is considered for all grid points, equidistant arranged along the
circumferential direction at meridional $m = 0.6$, between the hub and shroud
midway in the diffuser channel. Among these points one is chosen as a reference
located at twelve o’clock. Subsequently, the space-time cross correlation is
computed as:

$$R(r_i, \Delta t) = \frac{<u'(x_i, t)u'(x_i + r_i, t + \Delta t)>}{\sqrt{<u'(x_i, t)^2>}\sqrt{<u'(x_i + r_i, t + \Delta t)^2>}}$$  

(2)

The cross-correlation is then normalized and the result for flow angle cases
$\alpha = 9.5^\circ$ and $\alpha = 5.5^\circ$ are presented in Fig. 9. The fluctuating tangential
velocity component utilized in the space-time correlation is obtained by re-
construction of the most energetic Fourier surface spectra flow modes. The
most energetic modes are determined from Fig. 7. Thus, for flow angle case
Figure 9: Space-time cross correlation of the tangential velocity fluctuating component along the azimuthal direction at $m = 0.6$ and in the middle of the diffuser section at $n = 0.5$. a) flow angle $\alpha_{IN} = 9.5^\circ$, tangential velocity fluctuation reconstructed from Fourier surface spectra modes at $0.04f_{BPF}$ and $f_{BPF}$, respectively. b) flow angle $\alpha_{IN} = 5.5^\circ$, tangential velocity fluctuation reconstructed from modes at $0.03f_{BPF}$, $0.07f_{BPF}$ and $f_{BPF}$. The correlation is normalized, and the scale range is from -1 to 1.

$\alpha_{IN} = 9.5^\circ$ the contributions from tonalities at $0.04f_{BPF}$ and $f_{BPF}$, respectively, are used. For the lower flow angle case $\alpha_{IN} = 5.5^\circ$, the contributions from $0.03f_{BPF}$, $0.07f_{BPF}$ and $f_{BPF}$ are used. A coherent pattern is observed with several inclined lines exhibiting strong correlation. This is directly linked with the propagation of the rotating cells. The slopes of the inclined lines are approximately constant and correspond to the local convection speed. For the large flow angle case, there are two disturbances along the circumference, which correlates with the two rotating stall cell structures in Fig. 8, (i.e. $0.04f_{BPF}$ for $\alpha_{IN} = 9.5^\circ$). For the smaller flow angle case there are thus five disturbances, since there are five rotating stall cell structures. For $\alpha_{IN} = 9.5^\circ$ the local convection speed is approximately 50% of the impeller speed. Now, the slope of the characteristic is different for $\alpha_{IN} = 5.5^\circ$, which is approximately 30% of the impeller speed.

Another interesting observation in Fig. 8 is that the radial and tangential velocity fluctuation distributions elucidate streaky elongated flow features, which propagate with the rotating stall mode. The length scale of the streaks is in the order of the diffuser channel width. Close to the inlet these features are located in the viscous sublayer rather in the core flow, which correlate with the amplified Reynolds stress levels close to the wall as shown in Fig. 5. They
can be identified as features carrying turbulent energy, which is an important characteristic of wall-bounded turbulence. Further out in the radial direction these stream-wise streaks grow with increased interaction and mixing with the core flow, see Fig. 10. This partially motivates why the Reynolds stress distribution in the tangential direction becomes fuller in the mid of the diffuser channel.

\[
\alpha_{IN} = 9.5^\circ \\
0.04f_{BPF}
\]

\[
\alpha_{IN} = 5.5^\circ \\
0.07f_{BPF}
\]

Figure 10: Fourier surface spectra decomposition of the the wall normal velocity fluctuation component colored on the side view plane, between meridional stations \( m = 0.3 \) and \( m = 0.4 \). Constrained streamlines are overlayed for the corresponding to the flow modes \( \alpha_{IN} = 9.5^\circ \) at \( 0.04f_{BPF} \) and \( \alpha_{IN} = 5.5^\circ \) at \( 0.07f_{BPF} \), respectively.

Qualitatively, there is a resemblance with the stream-wise streaks found in the turbulent boundary layer in curved channel flow. However, in this case there is the added complication of an interaction with a rotating stall mode feature and an adverse pressure gradient. It is known that secondary flow instability can appear in a turbulent boundary layer over a curved channel wall. An important factor is when the boundary layer thickness is comparable to the radius of curvature. In such scenario, a centrifugal action creates a pressure variation across the boundary layer. This is said to lead to formation of longitudinal vortices (Saric 1994). In Fig. 10, for the selected meridional section, for the larger flow angle case \( \alpha_{IN} = 9.5^\circ \), two major rotating secondary flow structures are seen with counter clockwise rotation. These are located closer to the shroud wall. The other major vortex in between have a clockwise rotation and is located closer to the hub wall. Consequently, just ahead of the clockwise rotating vortex \( \overline{u_n} \) is positive and just behind it \( \overline{u_n} \) is negative. A similar sequence of updraft and downdraft of the wall normal velocity component between the hub and shroud walls can also be seen for the small flow angle case \( \alpha_{IN} = 5.5^\circ \). The onset of the instability can be estimated with the Görtler number, which is the ratio
of centrifugal to viscous forces in the boundary layer. For the considered diffuser geometry this number exceeds 0.3, which is the critical limit for occurrence of the Görtler instability. This is one possible explanation for the coherent streaky features found superimposed on the rotating stall mode (see Fig. 8).

4. Conclusions

The computation of the mean velocity components in a narrow vaneless diffuser at low flow angles suggests a possible connection between the tendency of boundary layer separation in an adverse pressure gradient and the tendency of flow circulation and propagation into a rotating stall consisting of counter-rotating flow structures. This is in agreement with published literature on the subject, e.g., Jansen (1964); Tsujimoto et al. (1996). For the diffuser geometry considered in this study a tendency of an inflection point was detected in the time-averaged meridional velocity profile close to the hub wall at the top of the turnaround location. This correlates with a sudden reduction in the near-wall stress level in the near-wall region, which is indicative of approaching boundary layer separation.

An emerging feature of the rotating stall instability is the formation of counter-rotating flow structures. Such features of the flow causes unsteady effects on the turbulence structures residing in the boundary layer. One important perturbation mechanism is linked with the pressure gradient developed and the impact of the streamwise curvature. This conclusion is associated with the peak Reynolds stress level moving closer to the core flow and the response in the near-wall hub region, where flow blockage occurs. Another interesting feature is the characteristic turbulent streaks found in the turbulent boundary layer, which are being convected with the rotating stall instability. Those are turbulent energy carrying features. They were found to form longitudinal vortices with length scale in the order of the diffuser channel width. These features share resemblance with the secondary flow instability in turbulent boundary layers subjected to curvature effect.

A strong shear-layer is located close to the inlet, which is due to the considered shape of the imposed boundary condition. This correlates with high Reynolds stress levels and is another possible mechanism for the onset of the rotating instability which is seen to grow further downstream in the diffuser. The point-to-point cross-correlation was computed in order to determine the phase differences between the dominant flow modes, i.e. the rotating instability mode and the blade passing frequency mode. This is understood to be a convolution over a finite period with similar notion to a Fourier transform with assumption of periodicity. Consequently, all notions of causality are lost and one can no longer refer to driver and response signals, but only signals which are correlated. Therefore, it is challenging to say which signal drives which. However, the numerical results shows a strong coherence between shear-layer instability close to the inlet and the developing instability prior to the turnaround location.
5. Acknowledgment

This work was conducted at the Competence Center for Gas Exchange (CCGEx). The Swedish National Infrastructure for Computing (SNIC) via HPC2N and PDC, the Parallel Computing Center at KTH, are acknowledged for providing the computing resources.

REFERENCES


