Doctoral Thesis

Flow structure and stability of a turbocharger centrifugal compressor

Author(s):
Schleer, Matthias Wolfgang

Publication Date:
2006

Permanent Link:
https://doi.org/10.3929/ethz-a-005214642

Rights / License:
In Copyright - Non-Commercial Use Permitted
Flow Structure and Stability of a Turbocharger
Centrifugal Compressor

ABHANDLUNG

zur Erlangung des Titels
Doktor der Technischen Wissenschaften
der
EIDGENÖSSISCHEN TECHNISCHEN HOCHSCHULE
ZÜRICH

vorgelegt von

Matthias Wolfgang Schleer
Dipl.-Ing. RWTH Aachen
geboren am 09. Mai 1973
in Freiburg im Breisgau, Deutschland

Angenommen auf Antrag von
Prof. Dr. Reza S. Abhari, Referent
Prof. Dr. Seung Jin Song, Korreferent
Dr. Beat Ribi, Korreferent

Zürich 2006
Acknowledgements

This thesis is a product of my research work at the Turbomachinery Laboratory at the Swiss Federal Institute of Technology Zürich (ETHZ). I would like to thank first and foremost Professor Dr. Reza S. Abhari for his dedicated support and guidance throughout the entire research project. I am most grateful not only for his encouragement and the continuous scientific discussions that drove this project - but also for his willingness to raise funds for the continual re-building of the lab's test rig.

Professor Dr. Seung Jin Song has kindly agreed to accept the role of co-examiner and has helped me to keep my focus on the relevant research questions. His gift of asking the appropriate questions at the right time helped to weave the story.

Special thanks belong to my other co-examiner, Dr. Beat Ribi, who has always pointed out the importance of combining research and industrial applicability. His support in designing the compressor stage and his broad knowledge in the field of centrifugal compressors is greatly appreciated.

During the research phase, I received many thought-provoking impulses by Dr. Anestis Kalfas. Dr. Martin Rose always had an open ear and a high interest in the thermodynamic aspects of the work. Professor Ndaona Chokani is thanked for his helpful introduction on high order spectral analysis. I would also like to thank Dr. Joël Schlinger for invoking my interest in the probe measurement technology.

I am very grateful to the comprehensive assistance I received form all my PhD colleagues in the lab. In spite of diverse research topics and many different cultural backgrounds, I could always rely on their support in most important professional and personal concerns. We shared an important time during which I also had the chance to learn a lot from their works focusing on turbines, CFD, wind-tunnels, cooling jets, and probes.

An important part of the work was done in co-operation with students who worked within the research project as a part of their studies. Their effort and dedication must not be underestimated and is greatly appreciated.
Den Herren Peter Lehner, Hans Suter, Thomas Künzle und Christoph Räber gilt mein besonderer Dank für Ihre stets hilfsbereite Art und die kompetente Erledigung aller technischen und mechanischen Herausforderungen. Without their hard work and practical experience the experimental work required for this thesis would not have been possible. They have seen many generations of PhD students coming and going during their careers and are still eager to help them with their profound practical knowledge.

Mr. Cornel Reshef has been most helpful in tackling all electronic challenges and sharing his expertise to an “electronic fresher”. Mrs. Marlene Hegner has always ensured that all the administrative hurdles were handled in time.

Finally, I would like to express my gratitude to my wife Barbara and my son Fabian who have helped to shift my focus regularly towards the bright side of life.

Zürich, June 2006       Matthias Schleer
# Table of Contents

Abstract........................................................................................................... VII
Kurzfassung.................................................................................................... VIII

1 Introduction and Background .................................................................. 1
   1.1 Centrifugal compressors: Application and requirements ................. 1
   1.2 Turbochargers in automotive applications ..................................... 2
   1.3 Unsteady phenomena in centrifugal compressors ............................. 4
   1.4 3D flows in compressors with tip clearance .................................... 5
   1.5 Special features of small turbomachines ....................................... 6
   1.6 Scope, goals and organization of the thesis .................................... 7

2 Experimental Set-up .......................................................................... 9
   2.1 Research facilities.......................................................................... 9
   2.2 Stage and impeller design............................................................... 13
   2.3 Investigated diffuser and tip clearance cases................................... 25
   2.4 Experimental techniques............................................................... 27
   2.5 Cylindrical co-ordinate system....................................................... 28
   2.6 Averaging of time-resolved data .................................................... 29
   2.7 Acquisition and processing of stage performance data ................... 31
   2.8 High frequency response static pressure plugs............................... 32
   2.9 Laser Doppler Anemometry (LDA) ................................................. 33
   2.10 Aerodynamic probe technology..................................................... 38

3 Stability of the Centrifugal Compressor System .................................. 47
   3.1 Definition of the unsteady phenomena ......................................... 48
   3.2 Characterization of the operating condition .................................... 56
   3.3 Stability of the compressor stage.................................................. 60
   3.4 Influence of the Reynolds number............................................... 66
   3.5 Influence of tip clearance width on the performance ..................... 67
   3.6 Influence of diffuser shape on the behavior and range.................. 69

4 Evolution of the Flow in the Vaneless Diffuser ..................................... 73
   4.1 Circumferentially averaged diffuser flow pattern ......................... 74
   4.2 3-dimensional flow features in the vaneless diffuser ..................... 77
   4.3 Vortical structure of the flow in the tip gap region ......................... 88
   4.4 Improved model for the description of the diffuser flow ................ 92
   4.5 Diffuser flow pattern at off-design flow rates ............................... 94
   4.6 Explanation for the changed flow structure at off-design flow conditions ... 106
   4.7 Variation of the relative tip clearance width .................................. 111
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>5</td>
<td>Influence of Tip Clearance Width on Stability</td>
<td>119</td>
</tr>
<tr>
<td>5.1</td>
<td>Experimental set-up and data evaluation</td>
<td>122</td>
</tr>
<tr>
<td>5.2</td>
<td>Steady pressure distribution</td>
<td>124</td>
</tr>
<tr>
<td>5.3</td>
<td>Blade passing related periodic pressure fluctuation</td>
<td>126</td>
</tr>
<tr>
<td>5.4</td>
<td>Non-deterministic deviation of the pressure fluctuation</td>
<td>131</td>
</tr>
<tr>
<td>5.5</td>
<td>Comparison of the pressure fluctuations for baseline and reduced clearance</td>
<td>135</td>
</tr>
<tr>
<td>5.6</td>
<td>Modelling of the tip clearance flow</td>
<td>141</td>
</tr>
<tr>
<td>5.7</td>
<td>Summary of the stability mechanism</td>
<td>146</td>
</tr>
<tr>
<td>6</td>
<td>Discussion and Conclusion</td>
<td>147</td>
</tr>
<tr>
<td>A</td>
<td>References</td>
<td>153</td>
</tr>
<tr>
<td>B</td>
<td>Frequently used symbols</td>
<td>161</td>
</tr>
</tbody>
</table>
Abstract

This compressor research project deals with the aerodynamic behavior and stability of small-scale highly loaded centrifugal compressors. These compressors are widely used in distributed power applications and in automotive turbocharging. In the course of this thesis the flow structure and the formation of the secondary flows have been experimentally investigated. The operating regime near the stall line at part speed operation is of special interest. Two facilities of different scales were developed and equipped with a similar compressor stage. The real-scale facility provides the baseline for the comparison of the system behavior. The use of the large-scale research facility provides the opportunity to measure accurately the structure of the flow in the diffuser. Optical velocimetry systems and time resolved pneumatic probes have been applied to a variety of diffuser configurations. For analyzing the stability of the compressor system the wall pressure fluctuations along the shroud and diffuser have been measured in a time resolved manner. Based on these time resolved pressure fluctuations post-processing methods have been developed to characterize flow instabilities shortly after their formation. They provide insight into the onset of instability and the role of the tip clearance flow on stall inception.

A special focus is put on the effects of low Reynolds numbers, high blade-loading, and large relative tip clearance on the flow structure. These effects are common features of compressors in small-scale applications and are responsible for the deterioration of compressor performance at small scales. In order to find dependencies of stability and performance upon the Reynolds number a parametric study is performed. Therefore, the compressor behaviour is measured at the same operational point but at modified inlet pressure. It is found that in the current machine the Reynolds number has little effect on the compressor performance.

To investigate tip clearance effects, flow velocity measurements have been performed at different diffuser and tip clearance configurations. Using this data the formation of the diffuser flow structure and its dependence on the flow through the tip gap is studied. The tip clearance width has a dominating effect on the formation of secondary flows in the diffuser. The flow structure in this highly loaded compressor does not comply with the classical Jet-Wake pattern. For increased clearance, the clearance flow is identified as an additional highly vortical feature. This additional pattern near the shroud destabilizes the compressor and deteriorates the performance. Based on these findings, a
modified flow model which includes tip leakage is proposed. This 3-zone flow model is more appropriate for the description of small-scale compressors with large relative tip clearance.

Kurzfassung


Um Messungen an geometrisch kleinen Anlagen zu ermöglichen, wurden zwei Versuchsanlagen verschiedener Grösse entwickelt und mit geometrisch ähnlichen Verdichterrädern ausgestattet. Die Anlage in Echtgrösse liefert die Basis für den Vergleich der beiden Anlagen während die geometrisch skalierte Anlage akkurate, zeitlich und räumlich aufgelöste Messungen ermöglicht. Das Stabilitätsverhalten des Kompressors wurde mittels zeitauflösenden Wandrauchsensoren untersucht. Im Weiteren wurden Analysetechniken entwickelt, um Instabilitäten frühzeitig zu erkennen und die Rolle der Strömung über die Schaufelspitzen zu erklären.

1 Introduction and Background

1.1 Centrifugal compressors: Application and requirements

Centrifugal compressors are widely used in a broad spectrum of industrial applications where they are utilized for compression or conveyance of gases. In transport applications radial compressors are used in the gas turbines of helicopters or turbochargers for internal combustion engines. Recently, the application of small radial compressors in distributed power generation and micro-turbines has been discussed. The operational characteristic of a centrifugal compressor is displayed schematically in Figure 1-1. The characteristic shows the relation of the volume flow rate and the pressure ratio of the compressor stage for constant impeller speed. Additionally lines of constant efficiency are shown. Towards low mass flows the operation regime is limited by unstable phenomena like surge or rotating stall.

Figure 1-1 Typical operation characteristics of a radial compressor

The requirements for small radial compressors used in automotive turbocharging and distributed power applications are ambitious. A wide operating range and high efficiency are required throughout the operation envelope. The challenge for further developments of centrifugal compressors is to broaden the operation range by shifting the stall line without changing the
maximum mass flow or the efficiency. The problems faced here involve complex 3D flow structures, loss generation mechanisms, impeller/diffuser interactions, and more general issues like the stalling mechanism and mechanical stress considerations. In addition, all solutions need to be cost effective as turbocharging becomes only feasible if low cost turbocharger are available. A comprehensive overview off the specific design process and requirements for compressors in different applications is given in a series of papers published in 1999 by IMechE (Came et al. [5], Dalbert et al. [13] and Flaxington et al. [26]).

1.2 Turbochargers in automotive applications

A turbocharger increases the air pressure at the inlet manifold of the engine. As the air pressure and thus the density is higher more oxygen can be drawn into the cylinder to burn more fuel. Unlike a mechanical supercharger a turbocharger does not feed off mechanical power form the engine. Instead, the turbocharger uses the waste energy from the exhaust gas to drive a turbine wheel that is linked to the compressor through a shaft. The benefit of turbocharging the engine is an up to 50% increase in power output for the same engine size. Altitude effects on the engine are compensated as air is delivered at an increased pressure to the inlet manifold. Fuel consumption and emissions are improved as the thermodynamic cycle is enhanced and the internal friction of the engine is reduced.

As well as the engine itself, the turbocharger is subject to continuous variation of the load demand. Especially in automotive applications the engine defined mass flow rate can vary widely from the designed best point. This trend is seen in Figure 1-2, where the demand of an internal combustion engine is plotted within a turbocharger delivery characteristic. The operation line at part load of the engine is very close to the surge line of the compressor. At full engine load the operation of the compressor is restricted by the choke line and the maximum impeller speed. These variations of the load demand an efficient and safe use of the turbocharger over a large range. The compressor must be able to operate over long periods of time at part load without any danger of instabilities or mechanical failures. One of the goals for further development of impeller blading for automotive turbocharging is a reduced sensitivity to loading changes and a broader operational range at an unchanged pressure rise.
1.2 Turbochargers in automotive applications

For turbocharging of automotive internal combustion engines usually mass flows rates below 0.1 kg/s are required at pressure ratios of up to 2.5 for gasoline and up to 3 for diesel applications. These demands are met by compressors with impeller diameters below 70 mm, which are operated at high shaft speeds in the order of 2,000 to 3,000 rotations per second. To improve the response on load transients, the polar moment of inertia and therefore the exit diameter has to be reduced further which leads to higher rotational speeds and higher material stresses. To maintain relatively low manufacturing costs cast impellers are often used. This restricts the impeller design to simple geometries and highly 3-dimensional and undercut designs are avoided. Usually splitter bladed designs with 5-8 blades and high backsweep angles are utilized. Due to the small size of the compressor larger relative tip clearances are seen than in large-scale industrial applications. As a special need of high volume production the automotive industry requires simple engineering solutions that allow for high performance as well as good reliability and serviceability at low costs. Therefore highly sophisticated solutions like adjustable vanes or control systems applied in aircraft engines may not be suitable for the automotive industry.

Figure 1-2 Demand of an internal combustion engine within the characteristic of a waste gate controlled turbocharger (Mayer [59])
1.3 Unsteady phenomena in centrifugal compressors

Towards small mass flow rates the operation of a compressor is limited by flow and system instabilities like surge and stall. In the worst case these instabilities are capable of damaging the compressor and its surroundings. To avoid operation in these regimes a profound understanding of the phenomena and their inception mechanism is needed for each specific compressor arrangement. In the open literature many reports are available on the phenomena of rotating stall and surge in axial and radial compressors (Emmons et al. [23], Jansen [44], Senoo and Kinoshita [75], Greitzer [31], Fink et al. [25], Longley [58]).

In all studies rotating stall is understood as a travelling non-axisymmetric distortion of the circumferential flow pattern. The structure of rotating stall is similar in all kind of machines and a phase shift of the pressure fluctuations can be found. The traveling speed of the flow distortion is a fraction of the impeller rotation speed. In vaned diffusers a single stall cell rotates at 20 to 70 % of the rotor speed within the diffuser. In unvaned diffusers, stall patterns with up to 5 stall cells have been reported. Each of these cells travels with the same speed as a single cell resulting in a pressure fluctuation with a higher frequency than the rotor speed.

Surge is understood as an instability of the whole system resulting in oscillations of pressure and mass-flow in anti-phase. The occurrence of rotating stall or surge at a given operation condition depends on the system layout and is described by the B-parameter (Greitzer [31]). This parameter gives a hint of which instability is likely to occur first when the mass flow is throttled to lower values. Active stabilization mechanisms of the compressor by unsteady air injection have been studied at NASA Glenn Research Center and at MIT (Tryfonidis et al. [93], Weigl et al. [95]). Based on the description in axial compressors Spakovszky [83] extended the analytical stability models on radial machines and showed an improvement in stall margin by using active control.
1.4 3D flows in compressors with tip clearance

For automotive turbocharging and small-scale distributed power applications unshrouded impellers are used. In these turbomachines the performance is degraded by the pressure loss and the formation of secondary flows in the tip gap region. The losses are caused by interaction of the flow through the clearance between the casing and the impeller blades. The effect of the tip clearance flows on small turbomachines is strong as the manufacturing accuracy determines a minimum clearance and thus the relative clearance gets large. As cast impellers are used the impeller design is restricted to designs with simple blade shapes and only few blades are used in order to ease the production of the impeller.

Details of the flow structure and losses were described in the past by a variety of authors (Dean [15], Traupel [90], Fowler [27], Eckardt [21&22], Denton [16], Johnson and Moore [48], Krain [53], Ibaraki et al. [39]) using either experimental or computational methods. Generally, the kind of blading used in small compressors shows a considerably high blade-to-blade loading and a strongly non-uniform flow. The radial velocities near the hub and the pressure surface of the blade are high while those near the shroud / suction side corner are low. The secondary flow mechanisms guide low momentum fluids to areas where they weaken the existing boundary layers and cause further separation. This mechanism is analogous to a wide-angle diffuser where separated flows are generated in the boundary layers. Current stage component design procedures are mostly based on quasi-three-dimensional design methods and ignore secondary flow features. Therefore these design methods could be improved only if the losses caused by the three-dimensional flow are considered.

Compared to vaned diffusers a vaneless diffuser usually provides a lower pressure recovery. But it is able to operate at a broader operating range and is much easier to build. Therefore they are used widely in automotive turbochargers where broad operating conditions are required but losses could be tolerated to some extent. Diffusion in vaneless diffusers is controlled by the angular momentum of the flow and the change of the cross sectional area with diffuser radius. The process is influenced by frictional viscous losses which alter the flow profile close to the diffuser walls. If the flow rate is decreased the flow entering the diffuser will become more tangential, the diffuser path will be longer and the frictional losses at the wall increase. Furthermore, under the adverse pressure gradient local overturning can occur near the shroud wall.
as in this region the radial flow velocity is lower due to the tip clearance. If throttled further this local overturning will then lead to stalling of the diffuser flow and later on to surge.

1.5 Special features of small turbomachines

Small turbomachines suffer from a significantly reduced operating range and efficiency compared to large-scale industrial machines. This is caused by a higher blockage and greater sensitivity to separation due to lower Reynolds numbers and geometry effects like simple blade geometries and larger relative tip clearance.

In 1955 Cordier [11] established the basis for the scaling of different machines by introducing a non dimensional set of parameters to describe the operation of turbomachines. Three years later Rotzoll [68] studied the behavior of a water pump under varying Reynolds numbers. He used water and oil at different temperatures to vary the density of the fluid at different rotational speeds. This approach allowed a variation of the Reynolds number in a range of three orders of magnitude. Out of his results he derived scaling factors for the pressure head and the internal machine efficiency. In 1972 Pampreen [61] investigated the influence of high altitude operation on the compressor efficiency of aircraft engines. As in high altitudes the density is reduced the Reynolds number is reduced as well. Pampreen showed that the Reynolds number can be used as a measure for the compressor performance at lowered densities at high altitudes. But he also concluded that for the scaling of compressor designs the effect of the relative tip clearance is important. The effect of relative tip clearance was also studied by Block et al. [4] who found an increased efficiency with decreased tip clearance.

In the 1980's various authors (e.g. Casey [7]; Strub et al. [88]) have performed investigations in order to develop correlations for the change of pressure head and efficiency of centrifugal compressors with Reynolds number. Simon and Büłskämper [80] extended the correlations to include the influence of the surface roughness on head and work coefficient. All these investigation have in common that they focus on the prediction of applications with a higher Reynolds number than the tested machine. Attempts have been made to apply the correlations for estimating the efficiency to very low Reynolds number applications like micro-gasturbines (Kang et al. [49]) even though for very small compressors the models are out of range and therefore may not be valid.
1.6 Scope, goals and organization of the thesis

Research objectives

Compared to large-scale industrial or aeronautical devices, the performance of small-scale compressors is inferior. This is evident in lower efficiencies and thus a reduced effectiveness of the system. In the scope of this experimental work, the aerodynamic mechanisms leading to the performance deterioration and the reduction in stable operating range are identified and quantified.

In particular, these research questions will be addressed:

• Are the existing models of the flow structure sufficient for the description of the flow pattern in the vaneless diffuser of a highly loaded compressor with a large relative clearance ratio?

• What are the effects of a large relative tip clearance ratio and a reduced Reynolds number on the stability and range of a centrifugal compressor?

• What are the interaction mechanisms of the flow features in the vaneless diffuser and how do they affect the stability of the compressor?

These research questions on the aerodynamic behavior of small turbocharger type compressors are answered based on experimental data. An enlarged research facility and a true scale model were equipped with a representative model of a centrifugal compressor used in automotive or distributed power applications. The enlarged facility provides the opportunity for detailed measurements while the true-scale model provides the baseline for the comparison. Time-resolved measurements of the static pressure along the shroud and the flow structure within the vaneless diffuser have been taken to identify relevant flow features. Based on the time resolved flow measurements, an improved description and a model of the flow in the vaneless diffuser are provided. Also, the stability of the compressor have been determined for a variety of operating conditions and mechanisms leading to the onset of instability have been identified.
Thesis organization

In the first chapter of the thesis, an overall description of the requirements and applications of centrifugal compressors is given. Further, a broad overview of the flow structure and stability is given.

In the second chapter, the experimental set-up and the impeller design are described. Additionally, in the second chapter a description of the adopted measurement techniques is given.

In the third chapter, the stability and behavior of the compressor system are determined by evaluating measured high frequency pressure traces. A description of the applicable instabilities and a methodology for their detection is presented. The frequency content of a measured pressure trace was used as an indicator for the stability of the compressor system. Operating maps and the stability behavior of different configurations have been acquired and are evaluated using these detection algorithms. In this evaluation, it is seen that the relative clearance has a dominating effect on the compressor performance.

In the fourth chapter, the flow structure within the vaneless diffuser is measured using time resolved measurement techniques. Based on these measurements, an improved model of the flow at the impeller exit is given. This model contains information on the tip clearance flow and the interaction of different flow pattern in the diffuser.

In the fifth chapter, the stability of the compressor stage is evaluated further. In the highly resolved measurements of the static pressure along the shroud the path of the tip clearance flows is clearly seen. It is found that interaction between the clearance vortex and the main flow is able to trigger instability. These measurements provide insight into the onset of instability and the role of the tip clearance flow on stall inception.

Finally, in a conclusion of the work and suggestions for further investigations is given.

The geometry of the investigated centrifugal compressor stage is available for further investigations by contacting the author or the Swiss Federal Institute of Technology, Turbomachinery Laboratory, Sonneggstrasse 3, 8092 Zurich, Switzerland.
2 Experimental Set-up

2.1 Research facilities

The measurements have been carried out on the centrifugal compressor facilities “Rigi” and “miniRigi” in the Turbomachinery Laboratory of the Swiss Federal Institute of Technology Zurich (ETH).

Both facilities are equipped with impellers of similar geometry and are operated with air in a closed loop. The “miniRigi” facility represents the scale of compressors used in automotive turbocharging or distributed power applications. The “Rigi” facility is scaled up by a factor of 1:6.06 to allow measurements with time-resolved flow and pressure measurement techniques. If all non-dimensional parameters are maintained the same flow phenomena will occur. Thus, experimental investigations of the flow field typical for small centrifugal compressors can be performed at a larger length-scale and detailed measurements become possible.

In both facilities the total temperature and static wall pressure at the inlet and outlet are measured to determine the stage performance. Static pressure taps are located along the flow path in the shroud casing and the diffuser back- and front walls. Both systems are equipped with flush-mounted high-frequency pressure transducers to characterize the system stability behavior.

“Rigi” facility

Detailed measurements using aerodynamic probes and optical Laser Doppler Anemometry (LDA) systems are carried out on the large scale research facility “Rigi”. The facility consists of a single stage, vaneless centrifugal compressor system operating in a closed loop. It is driven by a 440 kW direct current (DC) engine via a two-stage gearbox. The maximum shaft speed is limited to 22,000 RPM by the sealing system on the impeller shaft. The shaft speed is measured with a tachometer which is directly mounted on the shaft of the driving engine. The overall layout of the machine is shown in Figure 2-1.

The compressor is equipped with an centrifugal impeller of an outer diameter of D_2 = 400 mm. The rotor is followed by a parallel vaneless diffuser with an exit diameter of D_5 = 580 mm. It is designed for air and delivers a design volume flow rate of V = 3.5 m³/s at a design pressure ratio of π = 2.8.
The mass-flow rate is measured in the return duct with an orifice according to DIN 1952. The compressor is operated in a closed loop. The inlet pressure can be varied within a range of 16 to 125 kPa. Due to power limitations of the gearbox the maximum inlet pressure is limited to 60 kPa for operation at maximum shaft speed. The impeller load can be varied with a throttle downstream of the cooler. The closed loop arrangement facilitates measurements under repeatable and constant conditions without depending on the atmospheric conditions (pressure and temperature).

A flow straightener mounted in the suction pipe upstream of the impeller ensures axial flow at the stage inlet. The vaneless diffuser configuration consists of two parallel walls. The inlet and outlet diameters are 400 mm and
580 mm, respectively. A large toroidal collecting chamber follows the radial diffuser and ensures circumferentially uniform conditions in the diffuser arrangement under all through-flow conditions. Since the facility is operated in a closed loop, the compressed air has to be cooled using a counter flow air-water heat exchanger. The pressure is reduced to inlet conditions by the throttle and is controlled by a pressure regulation system.

In the large scale “Rigi” facility different measurement techniques are applied to get detailed information on the flow and pressure field at several positions within the stage. The Laser Doppler Anemometry (LDA) system is a commercial system available from TSI Inc. which is capable of measuring all three velocity components (Stahlecker [86]). The in-house developed Fast Response Aerodynamic Probe FRAP (Kupferschmied [54]) is used to determine the time-resolved static and total pressure, the velocity vector and the time-averaged stagnation temperature. For consistency check, pneumatic probes are also used. For acquiring the fluctuations of the static wall pressure high frequency pressure plugs are used at 30 positions along the shroud. A more detailed description of the measurement systems and the data processing tools are given in Sections 2.9 and 2.10.

“miniRigi” facility

To study and verify scaling effects an actual-scale compressor was built. This small scale compressor allows design variations at lower operating and manufacturing costs and provides the baseline for the investigation of scaling and Reynolds number dependent effects.

The “miniRigi” research facility is based on a 3-K Warner automotive turbocharger unit 1. In this facility the original impeller is replaced by an impeller which is geometrically similar to the one used in the “Rigi” research facility. The impeller is driven by the original turbocharger turbine rotor (Figure 2-2). The bearing and shaft seal arrangements remain unchanged. The turbine is supplied externally with compressed air from the shop air system at 400 KPa. The rotational speed is controlled by two throttle valves of different dimensions arranged in parallel in front of the turbine inlet. One of the valves, the larger one, is used in a stepping mode for large scale speed changes. The other one is smaller and is used to control the operation speed exactly to the desired value.

1. Turbocharger Type K26-2664GA6.91
For measuring the rotational speed, an optical sensor is placed on the turbine side of the turbocharger. The rotational speed can be set to values from about 20,000 RPM to 133,000 RPM. The flow rate of the compressor stage is controlled by an adjustable throttle which is positioned after the storage volume. It is measured with a standard orifice in the inlet pipe-work. To ensure axial flow, a flow straightener is mounted in the inlet suction duct. As the facility is operated in a closed loop the compressed air has to be cooled using a counter flow water heat exchanger. An autonomous oil circuit with an independent hydraulic pump lubricates the shaft and bearing arrangement. Further informations on the “miniRigi” facility and its layout are given by Schindler [71] and Germann [28].
2.2 Stage and impeller design

Design intent

In automotive turbocharging mass flows of below 0.1 kg/s are needed at a pressure ratio of 2.5 for gasoline and up to 3 for diesel applications. The working fluid is air at ambient inlet pressure and temperature. For these small mass flow rates and high pressure ratio applications impellers with outer diameters below 70 mm are used at rotational speeds of 2,000 to 3,000 rotations per second. This results in tip mach numbers on the order of $\mu_t = 1.3$ and higher. Usually designs with splitter blades and a low blade-count of only 6-8 main blades and blade backswep angles of up to 40° are utilized. To maintain relatively low manufacturing costs most impellers in automotive applications are cast from aluminium.

A compressor stage which meets the common design rules of automotive turbocharger compressors was designed and manufactured. To reproduce these general features the impeller was designed with a backswep angle of 30° from the radial and a blade-count of 7 full and 7 splitter blades. In both facilities a similar relative tip clearance is applied.

Scaling of the compressor stage

For the scaling of a compression stage the specific flow coefficient $\phi$ and the specific work coefficient $\psi$ have to be matched. The inlet conditions, here represented by the inlet Mach number $Ma_1$ and the non dimensional rotational speed $Mu$ have to remain constant.

\[
\phi = \frac{V_0}{D_2^2 \cdot U_2} \equiv const \tag{2.1}
\]

\[
\psi = \frac{\Delta h_{sTT}}{U_2^2} \equiv const \tag{2.2}
\]

\[
Mu = \frac{U_2}{\sqrt[\gamma]{\gamma RT_1}} \equiv const \tag{2.3}
\]
The impeller used as a baseline on the turbocharger facility “miniRigi” is a true-scale geometry following the design rules applied in automotive turbocharging. It shows an outer impeller diameter of $D_2 = 66$ mm and an exit width of $b_2 = 2.8$ mm. The design volume flow rate is set to $0.095 \text{ m}^3/\text{s}$ at a design pressure ratio of $\pi = 2.8$ and an impeller speed of 133,000 RPM.

In the enlarged “Rigi” facility a geometrically up-scaled impeller design is used. The scaling factor is 1:6.06 which results in an exit diameter of $D_2 = 400$ mm and an exit width of $b_2 = 17.1$ mm. Keeping the impeller tip speed $U_2$ unchanged the angular velocity $\Omega$ is lowered by the scaling factor in the enlarged “Rigi” facility. For maintaining a constant Reynolds Number the inlet density (i.e. the inlet pressure) has to be reduced by the scaling factor in the “Rigi” facility. This results in the need for a closed loop operating under vacuum conditions.

Under these conditions the axial inlet velocity component $c_z$ and the incidence angles on the blades remain unchanged for both compressor facilities. Assuming a similar stage efficiency and isentropic enthalpy rise, the design pressure rise $\pi_{Design}$ remains unchanged for both impellers. For a constant specific flow coefficient the volume flow rate is scaled by a factor of $(6.06)^2$. This results to a design flow rate of $3.5 \text{ m}^3/\text{s}$ for the enlarged facility. At 100 kPa inlet pressure this volume flow corresponds to a mass flow rate of 4.1 kg/s. The geometrical design parameters are summarized in Table 2-1. The inlet pressures for the ‘Rigi’ facility correspond to the pressure range where the motor and gearbox are able to deliver sufficient power for the maximum impeller speed. The range of Reynolds number represents the possible operation conditions for both facilities and is defined according Equation 2.5. For inlet pressures of 16 kPa in the “Rigi” facility and 100 kPa in the “MiniRigi” the same Reynolds number of $6.8e^4$ is obtained.
2.2 Stage and impeller design

**Design procedure**

The mean-line analysis and the 3D blade design have been performed using the commercial design package “Agile Engineering Design System” developed at Concepts NREC ([10]). The tool “COMPAL” was used to define the impeller and diffuser geometry based on models such as the Wiesner-Slip model and assumptions for the blockage and mixed out state. The 3D layout tool “CCAD” was used to define the hub and shroud contour, blade angles and blade thickness. After the preliminary design study, the results from the AGILE Design System were reproduced using a company code from MAN Turbo, Zurich. This was done to gain a compatible geometry for FEM and CFD analyses and to acquire a compatible input for the manufacturing routines.

**General layout and RZ contour**

Figure 2-3 shows the hub and shroud contours of the impeller and the inlet and diffuser arrangement for the enlarged research facility “Rigi”. The inlet and exit of the impeller are purely axial and radial, respectively. The inlet radius of the impeller is 35 mm at the hub and 106 mm at the shroud. The impeller exit

---

**Table 2-1 Summary of geometrical design parameter**

<table>
<thead>
<tr>
<th></th>
<th>miniRigi</th>
<th>Rigi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller exit diameter $D_2$ [mm]</td>
<td>66</td>
<td>400</td>
</tr>
<tr>
<td>Diffuser width $b_2$ [mm]</td>
<td>2.82</td>
<td>17.2</td>
</tr>
<tr>
<td>Tip speed $U_2$ [m/s]</td>
<td>460</td>
<td>460</td>
</tr>
<tr>
<td>Rotational speed [RPM]</td>
<td>133’000</td>
<td>22’000</td>
</tr>
<tr>
<td>Volumetric flow rate [m$^3$/s]</td>
<td>0.095</td>
<td>3.5</td>
</tr>
<tr>
<td>Inlet Pressure [kPa]</td>
<td>35 ... 130</td>
<td>16 ... 60</td>
</tr>
<tr>
<td>Mass flow rate [kg/s]</td>
<td>0.04 ... 0.15</td>
<td>0.6 ... 2.5</td>
</tr>
<tr>
<td>Reynolds number Eqn. 2.5</td>
<td>$2.5e^4$ ... $9.1e^4$</td>
<td>$6.8e^4$ ... $2.5e^5$</td>
</tr>
</tbody>
</table>
radius is 200 mm while the diffuser exit is at 290 mm. The diffuser was designed with straight and parallel walls as shown in the figure. In front of the impeller the hub is continued by a parabolic spinner which extends axially 55 mm in front of the leading edge. The geometrical dimensions are listed and non-dimensionalized by the impeller diameter $D_2$ in Table 2-2 for both facilities.

![Contour of the enlarged “Rigi” impeller](image)

**Figure 2-3**  Contour of the enlarged “Rigi” impeller

<table>
<thead>
<tr>
<th></th>
<th>miniRigi</th>
<th>Rigi</th>
<th>Normalized with $D_2$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet tip diameter $D_{IT}$ [mm]</td>
<td>35</td>
<td>212</td>
<td>0.53</td>
</tr>
<tr>
<td>Inlet hub diameter $D_{IH}$ [mm]</td>
<td>11.5</td>
<td>70</td>
<td>0.175</td>
</tr>
<tr>
<td>Exit diameter $D_2$ [mm]</td>
<td>66</td>
<td>400</td>
<td>1</td>
</tr>
<tr>
<td>Diffuser width $b_2$ [mm]</td>
<td>2.82</td>
<td>17.2</td>
<td>0.043</td>
</tr>
<tr>
<td>Diffuser diameter $D_5$ [m/s]</td>
<td>95.7</td>
<td>580</td>
<td>1.45</td>
</tr>
</tbody>
</table>
Tip clearance width

The relative tip clearance ratio is defined as the ratio between the tip gap and the blade height at impeller exit:

\[ CR = \frac{t}{b_2} \]  

(2.6)

In the experiments the baseline tip clearance ratio at impeller exit was set to a value of \( CR = 12.7\% \) in both machines. This large tip clearance ratio is common for small-scale applications as the axial displacement in the bearing is large compared to the diffuser width. In the small-scale application this clearance ratio corresponds to a tip gap of 0.35 mm while in the large-scale facility a tip of 2.2 mm applies.

In the large scale facility “Rigi” it is possible to reduce the axial tip clearance width at impeller exit to 0.7 mm by changing shim plates between the inlet assembly and the casing. The small tip gap corresponds to a clearance ratio of \( CR = 4.5\% \) and is equivalent to a well designed industrial scale compressor.

For the large scale facility the actual tip gap width along the meridional length of the impeller is displayed in Figure 2-4. The given tip clearance distributions were calculated for hot condition at full speed. The designed clearance during operation was verified using a novel optical measurement system (Kempe et al. [50]) which is based on low coherence interferometry. By use of this system it was proven that during operation and at hot condition the defined tip gap width appears.
Blade angle and thickness distribution

Figure 2-5 shows the distribution of the blade angles. All angles are given with respect to the meridional direction. The impeller is designed with a backsweep of 30° from the radial direction. The blades are designed straight without bow. To improve the part load behavior at low mass flows the design incidence on the blade leading edge is defined to be 3°. At the impeller inlet the blades are leaned 2° backwards while they have a 4° forward lean at impeller exit.

In Figure 2-6 the blade thickness distribution is shown. The splitter blades follow the main blades and are designed to match the thickness of the full blades after 35% of its meridional length. Compared to industrial designs of the same impeller diameter the blades are thick. This was done intentionally to scale the blade thickness of the cast impellers as they are used in automotive turbocharging.

Figure 2-7 shows the resulting 3D geometry as defined by the blade angle and thickness distribution. The nearly radial stacking of the blades is visible. An ellipse with a 1:5 aspect ratio defines the leading edges of the blades. For manufacturing purposes a fillet with a radius of 4 mm was introduced at the root of the blades.
Velocity triangle prediction at the design point

For this geometry the velocity triangles have been computed using the meanline design tool COMPAL for a volumetric flow rate of 4.1 m$^3$/s and a rotational speed of 22’000 RPM. The obtained velocity triangles are shown in Figures 2-8 and 2-9. At the inlet the triangles are given for the hub, mid height,
and shroud region while at the impeller exit the triangles are plotted for the primary and secondary zones. For impeller exit additionally the mixed out condition is shown. The mean flow velocities and angles are given for mixed out conditions and at mid height beside the triangles.

Figure 2-8  Velocity Triangle at impeller inlet

Figure 2-9  Velocity Triangle at impeller exit
Impeller cross-sectional area

Changes in the cross-sectional area are directly linked to the ratio of the relative velocities $w_2/w_1$. deHaller [14] proposed the ratio $w_2/w_1 = 0.75$ as an indicator for the risk of separation and blockage within the passage. By applying the simple geometrical relations:

\[
A_1 = \pi \cdot (R_{1T}^2 - R_{1H}^2) \cdot \cos \beta_1
\]
\[
A_2 = (2\pi R_2 b_2 - Nb_2 t) \cdot \cos \chi_2
\]

the inlet and outlet areas of the impeller can be calculated. For the given case, the inlet flow is assumed to be purely axial and the relative flow angle $\beta_1$ can be calculated. The relative flow angle $\chi_2$ is taken from the mixed out state in the velocity triangles in Figure 2-9. For the given design the ratio of the inlet and the exit area is close to unity. Under these circumstances the diffusion in the impeller passage is only a function of the density ratio, and the deHaller relation yields:

\[
Ha = 1 - \left( \frac{w_2}{w_1} \right)^2 \cong 1 - \left( \frac{\rho_1}{\rho_2} \right)^2
\]

By using the design values obtained from the velocity triangles in Figures 2-8 and 2-9 the deHaller number becomes $Ha = 0.45$. This fulfills the stability criterion proposed by deHaller [14] for axial blade rows (Cumpsty [12]).

Blade loading distribution

Figure 2-10 illustrates the calculated Mach number and the blade loading coefficient at the hub, shroud, and mid span. The blade loading coefficient is defined as:

\[
C_n = \frac{(w_{PS} - w_{SS})}{w_{mean}}
\]

The blade loading describes the difference in the relative velocities between the suction and pressure surfaces and sets it into relation to the mean relative flow velocity. The distribution shown in Figure 2-10 was calculated using the WUFLOW solver of MAN Turbo AG at the design condition.
The figure reveals an aft-loaded design with a very high blade loading near the trailing edge at 92% meridional length. The blade loading coefficient exceeds the value used for industrial designs by far. The very high value is a result of the high-work design with only 7 full and 7 splitter blades. The large relative tip clearance and the high blade-to-blade loading are expected to cause a strong tip gap flow and therefore strong secondary flow patterns.

**Finite element analysis of the Mises stress and free vibration analysis**

The mechanical integrity of the impeller was verified by a finite element (FE) analysis by Szwedowicz [89]. In the FE analysis the static Mises stresses have been calculated for the design speed of 22,000 RPM and uniform temperature of 140°C. The assembly loads are modelled by imposing an axial load of 50 kN and a tangential load of 1.25 kN on the key slot walls as a reaction on the torque during operation. The stress analysis revealed feasible stress levels for the blades. Locally a yield stress above the allowable stress of aluminium was found for the shaft-hub joint (Figure 2-11). As the high stresses occur on the nodes representing the fixed boundary condition for the FE analysis the impeller was considered to be safe.
A free vibration analysis was performed on a cyclic FE model for a rotational speed of 22,000 RPM and a uniform temperature of 140°C. The results are given in the nodal diameter curves in Figure 2-12. The first eigenfrequency is above the first to fifth engine order. Possible excitations at the 0th, 1st and 3rd nodal diameter are found for higher eigenfrequencies, but they do not affect the integrity of the impeller within the test environment.

Figure 2-12  Nodal diameter diagram for the impeller A8C Rigi
Manufacturing

Both impellers were manufactured using a 5 axis milling machine using a flank milling procedure. No special care was put on the hub surface and therefore the hub shows the typical marks of the milling tool. After the milling operations the impeller were tested at 115% overspeed for mechanical integrity and balanced to fulfill the balancing quality level $Q = 2.8$ according to VDI ISO 20601. The balancing bores and the marks of the milling process can be seen in Figure 2-13 where the turbocharger-scale impeller used in the “miniRigi” facility is shown.

Surface roughness of the impeller

After manufacturing the surface roughness was measured in both impellers using an optical non-contact laser profilometer (Autofokus-profilometer 2010, UBM Messtechnik). The profilometer system is based on the auto focus principle: A laser beam of approximately 1 µm diameter is directed onto the surface of the object. By changing the location of a collimator lens the relative distance between the surface of the object and the measurement head can be inferred. The profilometer system is equipped with a translation stage to allow a line scanning of the surface. Line scans with a scan length of 3 mm and 1000 measurement points per mm were performed and the surface roughness was evaluated according to DIN 4776 from these scans (Simons [81]). From these measurements a mean surface roughness of $R_{Z_{iso}} = 33 \mu m$ was calculated for the “miniRigi” Impeller. To diminish optical reflections the enlarged impeller used in the “Rigi” facility was painted with “Rucopur 2K-Eisenglimmerfarbe”.

Figure 2-13  Impeller A8C miniRigi
For the painted large-scale impeller, a mean surface roughness of
$R_{Z_{iso}} = 42 \mu m$ was measured. Using the surface Reynolds Number definition
which is based on the mean surface roughness instead of the diffuser width

$$Re_{Rz} = \frac{U_2 \cdot R_z \cdot \rho_1}{\eta_1} \quad (2.11)$$

is becomes obvious that the enlarged impeller was not sufficiently rough to
obtain a similar roughness Reynolds number in both scales.

### 2.3 Investigated diffuser and tip clearance cases

To investigate the influence of the vaneless diffuser on stability a variety of
diffusers have been designed. Measurements on the parallel wall diffuser
revealed a flow deficit on the diffuser front wall. In literature this flow deficit
is linked to the instability of the system and pinched diffuser are used to
accelerate the fluid at the shroud. In a second set of measurements the
influence of the axial diffuser width on the performance have been tested.

The four investigated cases are shown in Figure 2-14 and are summarized in
Table 2-3. Case (A) represents the setup found in automotive turbocharger and
is used as the baseline case for all investigations. The diffuser width is set to
$b_2 = 17.2$ mm and the tip clearance is $t = 2.2$ mm. This case is referred to as
the “STR 17.2”. Case (B) has a diffuser width of $b_2 = 15.7$ mm and a tip
clearance of only $t = 0.7$ mm. This tip clearance case is representative of large
scale industrial applications and is referred to as the “STR 15.7”.

To investigate the effect of the diffuser width on the compressor behavior an
additional parallel wall diffuser with a 1.5 mm step at impeller exit and a
diffuser width of $b_2 = 15.7$ mm was built. Case (C) shows the large tip gap of
2.2 mm like case (A) but the diffuser width is set to $b_2 = 15.7$ mm like case
(B). This is done by mounting a plate on the diffuser shroud wall. This results
in a step of 1.5 mm at the radius $R = 202$ mm. For these parallel wall diffusers
measurements showed that the flow is locally reversed at the diffuser shroud wall. The diffuser was modified according to the design criteria given by the
AGILE design code and a pinched diffuser with a $5^\circ$ pinch angle was
introduced (Case (D)). The pinch starts at the radius $R = 201$ mm and has a
slope of $5^\circ$. The pinched case is also showing the large clearance of 2.2 mm
but due to the pinch the exit width is only $b_5 = 11.2$ mm.
Table 2-3 Summary of investigated diffuser and tip clearance cases

<table>
<thead>
<tr>
<th></th>
<th>Straight ( b=17.2 )</th>
<th>Straight ( b=15.7 )</th>
<th>Step ( b=15.7 )</th>
<th>Pinched ( 5^\circ )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Diffuser type</td>
<td>parallel vaneless</td>
<td>parallel vaneless</td>
<td>parallel vaneless</td>
<td>pinched vaneless</td>
</tr>
<tr>
<td>Tip clearance ( t ) [mm]</td>
<td>2.2</td>
<td>0.7</td>
<td>2.2</td>
<td>2.2</td>
</tr>
<tr>
<td>Diffuser width ( b_2 ) [mm]</td>
<td>17.2</td>
<td>15.7</td>
<td>15.7</td>
<td>17.2</td>
</tr>
<tr>
<td>Diffuser width ( b_5 ) [mm]</td>
<td>17.2</td>
<td>15.7</td>
<td>15.7</td>
<td>11.2</td>
</tr>
<tr>
<td>Clearance Ratio ( CR = t/b_2 )</td>
<td>12.7%</td>
<td>4.5%</td>
<td>14%</td>
<td>12.7%</td>
</tr>
</tbody>
</table>

Figure 2-14 Investigated diffuser configurations
2.4 Experimental techniques

For designation of the stage performance the total temperatures and static wall pressures at inlet and outlet are measured. Static pressure taps are located along the flow path in the shroud casing and the diffuser back- and front wall. The facility is equipped with flush-mounted, high-frequency pressure transducers for detecting and characterizing the unsteady pressure fluctuations during rotating stall and surge. Vibration sensors are installed at two locations at the rear journal bearing and are connected to the emergency stop system of the facility. The humidity of the air is measured in the inlet duct.

![Measurement locations for global performance data](image)

For measuring time- and spatially resolved flow properties, velocities and flow angles across the diffuser width two measurement techniques were used:

- A 2-Sensor fast response aerodynamic probe (FRAP) measurement system, developed in house at ETH Zurich, provides time resolved static and total pressure as well as flow angles and Mach number. The fast response probe is supplemented with a pneumatic probe which delivers circumferentially averaged flow vectors (Kupferschmied et al. [54] and [55], Roduner et al. [66], Köppel et al. [52], Gizzi et al. [29], and Schlienger [73])
- A commercially available 3D Laser Doppler Velocimetry (LDA) system which provides local flow velocities (Stahlecker [85] and [86]).
2.5 Cylindrical co-ordinate system

In Figure 2-16 the cylindrical coordinate system used for the postprocessing of the data obtained by the LDA and FRAP measurement system is shown. The circumferential component is positive in the sense of rotation of the impeller. The axial component is positive in the flow direction at the inlet i.e. in the shroud-to-hub direction. This definition leads to a right-hand cylindrical coordinate system ($R\theta Z$-System). The signs of the three velocity components $v_r$, $v_\theta$, $v_z$ are positive in the direction of the coordinate systems.

All of the data are evaluated according to the angle definition given in Figure 2-17. All angles are measured from the radial direction. The sign of the angle is positive in the direction of rotation. For a time resolved measurement at one single location, the time axis is proceeding against the direction of rotation. Therefore, to resolve the circumferential axis in a quasi-steady mode out of the time-signal the sign of the axis needs to be adopted.
2.6 Averaging of time-resolved data

To calculate and compare integral values for time-resolved measurements different averaging techniques are used:

- Ensemble or phase-lock averaging
- Circumferential or time averaging
- Spanwise or hub-to-shroud averaging
- Field or stage averaging

The ensemble averaging technique is frequently used in turbomachinery for the postprocessing of time-resolved measurements in order to resolve the periodic flow structure. By using this averaging technique the deterministic fluctuation of the flow is separated from the stochastic and turbulent fluctuations. For ensemble averaging based on the rotor position a time series of data is taken starting at the same rotor-fixed trigger point. After measuring a sufficient number of rotations the ensemble average is calculated using:

\[
\tilde{x}(t_n, z) = \frac{1}{m} \sum_{i=1}^{m} x_i(t_n, z)
\]

In the current work the data is ensemble averaged using the signal of a once-per-revolution (OPR) trigger to resolve the rotor position. The rotor speed was
very steady and therefore an ensemble averaging could be done with a similar
number of samples for each averaging loop. In unstable conditions, an
additional time-base has to be introduced for each investigated flow instability.
The hub-to-shroud distribution of the flow quantities can be obtained from a
circumferential averaging. As the speed of the investigated machine is
constant the result is similar to a time-averaging of the flow quantities. The
time-averaging is described by the integral:

\[ \bar{x}_{(z)} = \frac{1}{t} \int x_{(t, z)} \, dt \]  \hspace{1cm} (2.13)

Circumferential or time averaging of a time-resolved measurement yields
results similar to those from a pneumatic probe. Köppel et al. [52] compared
the calculation of the circumferential average using the physical approaches by
Traupel [91] and Dzung [20] with the simple calculation as an arithmetic
average (Eqn. 2.14). They concluded that the usage of the simple formulation
of the circumferential or time-average as a the arithmetic average is sufficient.

\[ \bar{x}_{(z)} = \frac{1}{n} \sum_{i=1}^{n} x_{i(z)} \]  \hspace{1cm} (2.14)

If the measured data need to be compared with one-dimensional theoretical
predictions (Traupel [90]) or overall measurements of the stage performance,
the time-resolved data have to be reduced to global values of the flow
quantities. As the flow is severely skewed in hub-to-shroud direction, special
care is needed for averaging the different flow quantities (Roduner [65]). The
radial velocity has to be averaged using the area averaging approach:

\[ \bar{x} = \frac{1}{b} \sum_{0}^{b} x_{(z)} \Delta z \]  \hspace{1cm} (2.15)

while the work input, or the tangential velocity, has to be averaged using a
mass averaging approach:

\[ \bar{x} = \frac{1}{m} \sum_{0}^{b} x_{(z)} v_{r(z)} \rho_{(z)} 2\pi R \Delta z \]  \hspace{1cm} (2.16)
2.7 Acquisition and processing of stage performance data

In both facilities the operational data are acquired using a LabView based data acquisition program. In the data acquisition program DARLING, the overall stage performance is calculated from wall pressure taps and temperature probes in the suction and outlet ducts. The volume flow rate is measured using a standard orifice according to DIN 1952. The rotational speed of the impeller is measured by a tachometer at the power take-off of the gearbox. Within the software DARLING the voltage readings for all sensors are recorded and converted into physical values using 2-point calibration functions. Based on the measured data the operating conditions and non-dimensional characteristic numbers are calculated. The values are displayed continuously on the screen. To assemble a complete compressor map the data can be stored in a file. The current operational and performance data are provided to other LabView applications like the high frequency wall pressure or probe measurement program using a TCP/IP protocol. Figure 2-18 shows the front panel of the DARLING software. On the left hand side of the diagram the time history of the temperatures and pressures over the last 100 samples are displayed, while in the column on the right the actual operating condition is given. In the center an operating map is assembled from the saved data.

![Figure 2-18 LabView based data acquisition program DARLING](image-url)
2.8 High frequency response static pressure plugs

The unsteady pressure fluctuations in the diffuser are measured using flush-mounted pressure sensors with a high temporal resolution. Highly time resolved pressure plugs were developed in-house and are successfully applied. In order to check the measurements piezo-electric pressure transducers ² were used and are found to provide similar results.

The in-house developed pressure plugs are assembled using a 1 bar differential high sensitive piezo-resistive pressure transducer ³. The reference pressure side of the transducer is connected to a static pressure tap along the flow path. The signals are amplified with a standard FRAP-amplifier with a cutoff frequency of 25 kHz. The data are sampled at 100 kHz to obtain a good time base for determination of the time shifts between two signals. The voltage signals delivered by the amplifier are converted into pressure differences using an offset-gain transformation function. The offset and gain values are determined for each sensor separately before the measurements by applying a known pressure on the sensors. For the on-site calibration in “Rigi” the reference side of the pressure plug is opened to atmosphere and the closed loop is pressurized.

Figure 2-20 shows the locations of the pressure sensors used in the large-scale research facility. The pressure transducers are located along the shroud of the impeller and within the diffuser. In the diffuser, several sensors are placed at different angular positions. In Section 3.2 measurements at positions P9 are investigated in detail for the detection and characterization of rotating stall. In a concluding measurement of the phase averaged fluctuation of the static pressures along the shroud the number of pressure sensors was increased resulting in an average spacing of 5 mm between the pressure sensors. In the small scale rig time resolved pressure sensors are installed in the diffuser at different angular positions on two radial locations only.

2. Kistler, 6052 A/D
3. Intersema Sensoric SA, Type MS 761-D
2.9 Laser Doppler Anemometry (LDA)

Experimental set-up

The flow velocities within the diffuser have been measured with a 3D commercial LDA measurement system available from TSI. The LDA system is a non-contact measurement system to measure optically the velocity of seeding particles in a flow. Figure 2-21 shows the basic setup of the optical LDA arrangement. In the current measurements a cross scatter arrangement and an absorption glass in the diffuser back wall was used to provide a sufficient data rate.

The flow was seeded with polydisperse liquid paraffin oil particles with a mean particle size of 0.4 µm. The proportion of the particles above 1 µm in diameter is negligible. The seeding particles have to be small enough to follow the motion of the air flow but still need to scatter a portion of the incoming light. In previous studies by Stahlecker [86], it was concluded that the particles used will experience only little slip relative to the flow but provide a sufficient scattering of the light. Flow velocities are sampled randomly whenever a particle is crossing the measurement volume. For each traversing point 40,000 data points have been stored together with the impeller positions obtained from a once-per-revolution trigger.
**Measurement principle**

Figure 2-22 shows the fundamental principle of an LDA system. For each velocity component two coherent single-wavelength laser beams are crossed in the measurement volume. The size of the measurement volume is only 100x100x500 µm. In the measurement volume, constructive and destructive interference form a pattern of bright and dark fringes. The distance of the fringes is only a function of the half angle between the crossing coherent laser beams and the wavelength of the laser light. A particle passing these fringe pattern scatters the light when it is in a bright fringe. The scattered light is collected in the receiving optic and is transformed to an electrical signal in the photo multiplier.

The theoretical signal of one particle crossing the measurement volume is given by a modulated waveform. As the fringe spacing is known from the optical setup this waveform carries information on the velocity component in the plane of the laser beams. Therefore the distance $d$ in the signal corresponds to the velocity component of the particle. If this procedure is done for all three velocity components a full 3D velocity field can be computed without the need for calibration of the signal reading. By using a Bragg-cell the direction of the fluid particle can be determined. A more detailed description of the measurement principle and its application to centrifugal compressor is given by Stahlecker [86] and Durst et al. [19].

*Figure 2-21  Experimental set-up of the LDA system (from Stahlecker [86])*
2.9 Laser Doppler Anemometry (LDA)

In “Rigi” the velocities are acquired in a cross-backscatter mode providing measurements close to the wall within the vaneless diffuser space. A detailed view on the set-up and alignment of the LDA optical system is shown in Figures 2-23 and 2-24. One LDA probe is set-up perpendicular to the window 4, while the second probe is arranged at an angle of $\Theta = 34.7^\circ$ to enable cross-scatter scanning 5 and thus an acceptable data rate is achieved. Special care has to be taken to correct for the misalignment due to the optical displacement of the quartz window. To achieve coincident measurement volumes, the relative alignment of both optical probes has to be checked using a microscope lens while the quartz window is installed. To suppress noise from reflection on the back wall a filter glass 6 has been inserted into the diffuser back wall. A similar setup has already been applied and tested successfully by Stahlecker [86].

---

4. Glass quality: Heraeus Suprasil 2
5. For postprocessing and improved accuracy an angle of $90^\circ$ would be optimal while inclination angles below $30^\circ$ should be avoided as the transformation errors get large.
6. Filter glass quality: BG3
To ensure optical access and avoid collisions with the rig the complete probe arrangement had to be turned by an additional angle of 10.4 degrees relative to the axial coordinate of the cylindrical impeller coordinate system (dotted lines in Figure 2-24). The vectors indicate the velocities as they are detected by the LDA system.

**Evaluation of the LDA data**

The LDA system is operated in a virtual 3D-mode where the complete 3D flow vector is reassembled from consecutive measurements. This became necessary to achieve an acceptable data rate, as the data rate for coincident measurements was much too low. The reasons for the very low data rates in the coincident mode include an uneven distribution of the laser power at different wave lengths, the optical arrangement of the sending and receiving optics, the scattering of the light by the particles, and differences in the photomultiplier. In the virtual mode, data rates between 300 and 15,000 Hz have been achieved for all flow directions throughout the complete flow field. Generally lower data rates are observed close to the diffuser back wall and for high rotational speeds. 

---

7. The data rate for 3D coincident measurements is only 1 to 50 Hz
8. Mu > 0.6
2.9 Laser Doppler Anemometry (LDA)

For each measurement location and velocity component 40,000 velocity samples are recorded. As the first step in the postprocessing of the data, the velocities are phase locked averaged according to the impeller position which was saved together with the velocity information by the LDA system. A phase locked average of the measured velocity information is calculated for one main channel and one splitter channel assuming similar flow for each main/splitter passage. In the phase averaging procedure 180 increments covering one main and splitter channel are computed. This is equivalent to a circumferential resolution of 0.3° or a spatial resolution of 1.1 mm at a radius of R = 210 mm. The phase average of each class is calculated using 100 to 300 data samples which ensures an appropriate statistical accuracy. The standard deviation of each class is in the order of 5 to 20 m/s. The highest standard deviations are found behind the trailing edge of the blades where highly vortical blade wakes are shed. Next a coordinate transformation is performed on the phase average data. First, the axial velocity is calculated as a linear combination of the “blue” and “violet” LDA components:

\[
v_z = \frac{v_{\text{blue}}}{\tan \Theta} - \frac{v_{\text{viol}}}{\sin \Theta}
\]  

(2.17)
Finally a coordinate transformation is done on the “blue” and “green” velocity information to yield the $v_r$ and $v_\theta$ velocity components in the impeller coordinate system. Together with the knowledge of the measurement location and rotational speed of the impeller the flow angles, and relative velocity components are computed and the complete velocity triangles can be drawn.

### 2.10 Aerodynamic probe technology

Two kinds of aerodynamic probes have been used for the research described in this thesis:

- Pneumatic multi-hole probes for measuring time averaged flow properties across the diffuser width
- Fast response aerodynamic probes (FRAP) for measuring time resolved flow across the diffuser.

In the investigated configurations, no downstream structures like diffuser vanes are present. Therefore it is sufficient to perform one single axial traverse for each operating condition. For the pneumatic probe, a circumferentially averaged set of flow angles and pressures is achieved. In steady and periodically unsteady flows the time resolved FRAP probe provides total and static pressures and an averaged recovery temperature. Using this information, the local Mach number and thus the 3-D velocities can be computed in a time-resolved manner. The periodic fluctuation of the flow caused by the blade passing can be resolved for measurements with the FRAP probe. For measurements with the pneumatic probe a 4-Hole probe was used and provides a time-averaged 3D flow vector within one measurement.

The calibration and application of the in-house developed FRAP technology were described in detail by a variety of authors and publications over the last decade. A comprehensive overview on the FRAP technology is given by Kupferschmied et al. [54] and [55]. Roduner et al. [66] describe the application of the FRAP technology on the “Rigi” centrifugal compressor facility while Köppel et al. [52] describe the postprocessing of the data. Gizzi et al. [29] compare the results obtained by the FRAP technology with LDA results. A description of the virtual 4-Sensor mode for measuring all velocity component independently and the post-processing software HERKULES is given by Schlienger [73]. Within this section only a short summary of the probe technology and its application to the current application is given.
Geometry and mode of operation a 2-sensor FRAP probe

Figure 2-26 shows the geometry of the probe used for all time resolved and pneumatically averaged probe traverses. The silicon pressure sensors are directly placed in the tip of the probe allowing for a very high aerodynamic bandwidth. This probe shape was developed by Schlienger [74] and was successfully applied in an axial turbine facility.

Figure 2-26  Layout of the 2-sensor FRAP probe

The 2-sensor FRAP probes are used in a virtual 4-sensor mode (Figure 2-27). In the virtual 4-Sensor mode the flow is measured by the probe in three subsequent measurements. In the first measurement the probe center hole is aligned with the mean flow direction. Next the probe is rotated in yaw by 42 degrees clockwise and counterclockwise, respectively. The data chains obtained for the three yaw positions simulate the pressure signals P1, P2, P3 and P4 that would be delivered by a 4-sensor probe with sensors at the same yaw positions.
The mean level of the signal represents the overall pressure level and provides information on the flow direction. The fluctuating part of the signal is composed of deterministic (rotor position determined) and stochastic (e.g. turbulence induced) pressure fluctuations. Phase-locked sampling based on the rotor position and ensemble-averaging techniques statistically eliminate all stochastic fluctuations. Thus, the deterministic fluctuations can be obtained for one rotation of the impeller. To achieve a statistically significant number of events, the sampling rate of the data acquisition system was set to 200 kHz. For all operating conditions 600k samples corresponding to 660 revolutions at a rotational speed of 13220 RPM were recorded.

**Calibration and modelling**

In turbomachines the mean value as well as the perturbation of velocity or pressure fluctuations are crucial for the accuracy of the measurement. Therefore, special care has to be put on the calibration and modeling of the FRAP probe system. First, a sensor calibration is performed. In this calibration step the silicon sensors embedded in the probe are calibrated for pressure and temperature in a environmental chamber. The sensors within the head of the FRAP probe are fed by a constant excitation current $I_e$ in a Wheatstone bridge circuit. Figure 2-28 shows that the output signal voltage $U$ depends strongly on the pressure applied and weakly on the temperature of the sensor. On the other hand, the excitation voltage $U_e$ is weakly dependent on the pressure but
strongly on the temperature. As temperature and pressure are not completely de-coupled a numerical calibration model is used for the transformation of the signal and excitation voltages into physical pressure and temperature readings.

Next, an aerodynamic calibration is performed in a defined flow using the free-jet calibration facility (Figure 2-29). Using this calibration the measured temperature and pressure fluctuations can be transformed into aerodynamic flow properties. In the free-jet facility the probe is immersed into a flow with a well defined Mach number and direction and turned automatically in the yaw and pitch plane. For use in the compressor facility “Rigi” the aerodynamic calibration has been performed at seven Mach numbers in the range between Ma = 0.2 and Ma = 0.6. For pneumatic probes it is sufficient to perform only an aero-calibration as no transformation between voltage readings and pressures is required.

![Figure 2-28 Typical output of the sensor calibration](image)

![Figure 2-29 Free-jet calibration facility](image)
For use in the postprocessing of the measurements the probe is modelled by parametric calibration models. In these models the flow around the probe is approximated by a surface spanned by the discrete data \( K_{(Ma,\phi,\gamma)} \) known from the calibration. Using the approximation of Weierstrass the calibration surface is approximated by a higher-order bi-variable polynomial:

\[
P(x, y) = \sum_{i=0}^{m} \sum_{j=0}^{n} k_{i,j} x^i y^j
\]  

(2.18)

The polynomial calibration coefficients \( k_{i,j} \) used within this approximation function \( P(x, y) \) are determined using a least-square method from the calibration data. In the calibration of the probe used in the current work usually polynomials of the 3rd order are used for the calibration of the sensor while 6th order polynomial functions are used in the aerodynamic calibration.

In Figure 2-30 the polynomial bi-variable calibration model of the 2-Sensor FRAP probe is shown. The probe was calibrated for a range of \( \pm 28^\circ \) in yaw and \( \pm 15^\circ \) in the pitch angle. The model for the calibration coefficient \( k_\phi \) is only dependent on the yaw angle \( \phi \) and is thus decoupled from changes in the pitch angle \( \gamma \). The model for the pitch angle dependent calibration coefficient \( k_\gamma \) is weakly dependent on the yaw angle \( \phi \). The calibration coefficients for the pressure dependent coefficients \( k_\varsigma \) and \( k_t \) are not ideally symmetrical around the \( \phi = 0^\circ \) position. This indicates a small offset in pitch angle which is compensated by the calibration model.

The standard deviation of the calibration coefficients is given in Table 2-1. The accuracy of the calibration model is similar to the accuracy stated by Schlienger [73] for pneumatic 4- and 5-Hole probes and dual 1-Sensor probes.

<table>
<thead>
<tr>
<th>Table 2-4</th>
<th>Model accuracy for the 2-Sensor FRAP probe</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Yaw angle</td>
</tr>
<tr>
<td>Accuracy</td>
<td>0.1 [°]</td>
</tr>
<tr>
<td>Error (% of dynamic head)</td>
<td>0.6 [%]</td>
</tr>
</tbody>
</table>
Postprocessing of measured data

As a first step in the postprocessing of the measurements in the compressor “Rigi” the measured voltages $U$ and $U_e$ have to be transformed into pressure and temperature using the polynomial model:

$$\Delta p = \sum_{i=0}^{m} \sum_{j=0}^{n} k_{p(i,j)} U^i U_e^j \quad (2.19)$$

$$T = \sum_{i=0}^{m} \sum_{j=0}^{n} k_{T(i,j)} U^i U_e^j \quad (2.20)$$

This model is applied separately for each sensor and uses the polynomial calibration coefficients $k_{p(i,j)}$ and $k_{T(i,j)}$ found during the sensor calibration.
As a second step in the postprocessing the flow vector is calculated from the pressures $p_1$, $p_2$, $p_3$, and $p_4$ acquired in the virtual 4-Sensor mode by subsequent measurements (see Figure 2-27). For this step non-dimensional groups $K_{(Ma, \varphi, \gamma)}$ of the measured pressures have to be assembled based on the time resolved measurement of the pressures $p_1$, $p_2$, $p_3$, and $p_4$ and the total and static pressure $p_t$ and $p_s$ for each time step. The flow vector is described by the pitch sensitive coefficient:

$$K_{\varphi(Ma, \varphi, \gamma)} = \frac{p_2 - p_3}{p_1 - \frac{p_2 + p_3}{2}} \quad (2.21)$$

and the yaw sensitive coefficient:

$$K_{\gamma(Ma, \varphi, \gamma)} = \frac{p_1 - p_4}{p_1 - \frac{p_2 + p_3}{2}} \quad (2.22)$$

From these non-dimensional coefficients the time resolved pitch and yaw angle $\varphi$ and $\gamma$ of the flow approaching the probe can be calculated using the calibration coefficient $k_{i,j}$ as:

$$\varphi = \sum_{i=0}^{m} \sum_{j=0}^{n} k_{\varphi(i,j)} K_{\varphi}^i K_{\gamma}^j \quad (2.23)$$

$$\gamma = \sum_{i=0}^{m} \sum_{j=0}^{n} k_{\gamma(i,j)} K_{\varphi}^i K_{\gamma}^j \quad (2.24)$$

Once the flow direction is known, the time-resolved total and static pressures $p_t$ and $\gamma$ of the flow can be derived from:

$$K_{t(Ma, \varphi, \gamma)} = \frac{p_t - p_1}{p_1 - \frac{p_2 + p_3}{2}} \quad (2.25)$$
2.10 Aerodynamic probe technology

\[ K_{s(Ma,\phi,\gamma)} = \frac{P_i - P_s}{p_1 - \frac{p_2 + p_3}{2}} \]  \hspace{1cm} (2.26)

and the polynomial expressions:

\[ K_{t(Ma,\phi,\gamma)} = \sum_{i=0}^{m} \sum_{j=0}^{n} k_{t(i,j)} \varphi^i \gamma^j \]  \hspace{1cm} (2.27)

\[ K_{s(Ma,\phi,\gamma)} = \sum_{i=0}^{m} \sum_{j=0}^{n} k_{s(i,j)} \varphi^i \gamma^j \]  \hspace{1cm} (2.28)

By solving these equations the total and static pressures are obtained. As a final step in the postprocessing a time-resolved flow Mach number is calculated from the time resolved total and static pressure:

\[ Ma = \sqrt{\frac{2}{\kappa - 1} \left( \left( \frac{P_i}{P_s} \right)^{\frac{\kappa}{\kappa - 1}} - 1 \right)} \]  \hspace{1cm} (2.29)

and the time-resolved 3D-flow velocities can be computed.

**Set-up and measurement location of the FRAP system**

The probe traversing system was mounted on the front wall of the diffuser at the radial location of 105% impeller exit radius. The probes are traversed axially across the diffuser width and are turned around the probe axis by a numerically controlled traversing system. This set-up allows measurements across the diffuser width (Figure 2-31). However, the position of the pressure tap limited the spatial extent of the measurements in close proximity to the diffuser walls. Valid ensemble averaged sets of measurements are obtained for a probe traverse from axial position \( z = 3.0 \) to \( 15.0 \) mm in intervals of \( 0.5 \) mm. The position of the impeller was determined from a trigger on the impeller shaft that provided a once per revolution trigger signal. This signal was stored together with the flow data.
Automated traversing algorithm

The time resolved FRAP probe was operated at every traverse position in a virtual-4-sensor mode. The fluctuating flow was measured in a time-resolved manner at three angular positions of the probe shaft. Due to the skew of the flow the average flow angle is changing as a function of the axial position within the diffuser. Therefore it was necessary to adjust the mean set angle of the probe for each measurement location. This was done using an automated AUTOFRAP algorithm which was implemented in LabView. The algorithm requires only the first set angle while no information on the expected yaw angle distribution is required before a measurement is started. For each subsequent measurement location the probe is set to the averaged angular position of the previous location. In each position the sensor signal was measured and stored for all three yaw positions required for the virtual-4-Sensor approach. This algorithm was used for pneumatic probe and FRAP probe measurements and ensured that the flow is approaching the probe always within the valid range given by the probe calibration model.

Figure 2-31 Measurement location for the probe measurement system
3 Stability of the Centrifugal Compressor System

During operation of a compressor, flow and system instabilities like surge and stall can occur. In the worst case these instabilities are capable of completely damaging the compressor and its surroundings. Compressors can only be operated in regimes where no danger of instabilities is apparent. Towards high mass flows the operational range of a compression system is limited by the system resistance or choking of the smallest flow area. In the smallest flow area or throat sonic flow conditions do not allow a further increase of the mass flow. At low mass flows the operation is limited by the onset of rotating stall and surge.

In dynamic applications, e.g. in turbochargers of automobiles, compressors often operate in a part load condition and near the stability limit as its operation has to follow the load demand of the engine. Figure 1-2 shows a typical characteristic of an automotive turbocharger and the demands of an internal combustion engine with gate control. The compressor operates at a variety of impeller speeds and particular at part load very close to the stability line. Therefore, a profound understanding of the phenomena and on possibilities to avoid them is needed.

Within this section the compressor performance is described and compared for different diffuser configurations. First, a literature survey and a definition of the unsteady phenomena in pumping systems is given. Based on these theoretical descriptions an automatized method for the characterization of the operating point is developed for the compressor stage and implemented in the experimental set-up. Finally a parametric study on the influence of the tip clearance ratio and the diffuser shape on the behavior and range of the compressor is presented.

In the parametric study the strongest influence on the stability and behavior is observed for variations in the relative tip clearance. Modifications of the diffuser shape show only limited enhancements on the compressor performance and usable map width. Therefore, the configurations with a parallel wall diffuser but different relative tip clearances are selected for detailed investigations of the flow pattern at the impeller exit.
3.1 Definition of the unsteady phenomena

General concept of the stability of a pumping system

The stability of a pumping system depends on the interaction of the compressor and the downstream plenum and throttle or load. In steady state operating points the flow rate and the pressure rise set by the compressor equals the flow demand and the pressure drop due to the load and friction. In static unstable operating conditions small perturbations either of mass flow or pressure demand cause the system to depart even further from the initial operating point. This static instability occurs if the slope of the throttle line is tangent to the pumping characteristic of the compressor. Another mode of instability is the dynamic unstable conditions where small perturbations are not sufficiently damped by the system. Dynamic instability is usually found in regions where the slope of the pumping characteristic is positive, i.e. the operating point is left of the maximum pressure rise.

A dynamically unstable pumping system can be considered as a mass-damper system as sketched in Figure 3-2. Using this analogy a lumped parameter model of the pumping system can be set up. The damping of the system determines if the energy within a small perturbation is amplified or is dissipated. The spring and damper constants depend on the setup and size of the components in the pumping system. The pump can provide negative damping to the system and is therefore e.g. during rotating stall able to add energy into system transients. Analyses of this kind of systems (e.g. Emmons et al. [23] and Greitzer [31]) indicate that the pumping system will be dynamically unstable beyond the peak of the compressor characteristic. However, depending on the geometric size and the throttle characteristic the onset of the instability can be different.

Figure 3-1 Instabilities in pumping systems (from Greitzer [31])
3.1 Definition of the unsteady phenomena

The interaction mechanism causing dynamic instability is illustrated in Figure 3-3: If the pumping system is disturbed by a small perturbation of the volume flow $\delta V$ the system is reacting with a small change in pressure rise $\delta \Delta p$. The resulting integral of the curve $\delta V \times \delta \Delta p$ is indicating either an input or dissipation of energy in the dynamic system. In stable operating conditions a negative integral is found which is indicating that the excitation energy is dissipated and the perturbation diminishes. In the case of a dynamically unstable condition the integral $\delta V \times \delta \Delta p$ is positive. The system is excited by small perturbations of the volume flow and energy is added to the system. By this added energy the perturbations are amplified and the system gets unstable. The amplification of small perturbations is associated with dynamic instability and can be found in all pumping systems.

Figure 3-2 Mechanical analogy of a simple pumping system (Greitzer [31])

Figure 3-3 Physical mechanism of dynamic instability (from Greitzer [31])
Surge

One of the dynamic instabilities occurring in compressors is surge. Surge is understood as a filling and emptying of the volume on the high pressure side. During surge an instability of the whole compressor system results in an oscillation of pressure and mass-flow in anti-phase. The amplitude of the mass flow oscillation is so large that flow reversal occurs. A schematic of the compressor system and a surge cycle is show in Figure 3-4. The cycle starts at (1) where the flow becomes unstable. Consequently, the pressure cannot rise and the compressor jumps to the reversed flow characteristics (2) where flow reversal occurs. As the pressure diminishes the compressor follows the negative branch of the characteristics to approximately zero mass flow (3). Next it jumps to (4) where it then follows the compressor characteristic to (1) and the cycle repeats. Further explanations of surge dynamics and modelling are given by Greitzer [31], Fink et al. [25], Longley [58] or Gravdahl and Egeland [30].

As mentioned above the onset of the instability depends on the sizes of the components as they will define the damping and spring coefficients. Based on the lumped parameter mass-damper model Greitzer defines the B-parameter:

$$ B = \frac{U}{2\omega_H L_c} = \frac{U}{2a_k} \sqrt{\frac{V_p}{A_c L_c}} \quad (3.1) $$

where $U$ is the rotor tip speed, $\omega_H$ is the Helmholtz resonator frequency, $a$ is the velocity of sound, $L_c$ is the effective duct length, $V_p$ is the volume of the

---

**Figure 3-4**  Compressor characteristic with surge limit cycle

As mentioned above the onset of the instability depends on the sizes of the components as they will define the damping and spring coefficients. Based on the lumped parameter mass-damper model Greitzer defines the B-parameter:
Definition of the unsteady phenomena

downstream plenum and $A_c$ is the through flow area (Figure 3-2). This parameter defines whether a fluctuation of the pressure delivery will affect the stability of the overall system or if the excitation can be damped elsewhere in the system. Cumpsty [12] suggests to rewrite the B-parameter as:

$$B = \frac{\left(\frac{\rho U^2}{2}\right)A_c}{\rho A_c L_c U \omega_H}$$  \hspace{1cm} (3.2)

This parameter can be explained as the relation between the pressure rise in the compressor and the forces required to produce small oscillations of the mass flow in the duct at the natural frequency $\omega_H$. The B-parameter can therefore be understood as the ratio of the compressor pressure rise and the pressure rise required to induce mass flow oscillations. Based on the B-parameter it can be determined which kind of instability is to be anticipated to occur first when throttling the mass flow to lower values. If the parameter B is sufficiently large surge occurs. If it is sufficiently small the perturbation will be damped and the overall system remains stable even if the compressor itself is unstable. In this case rotating stall either within the impeller or diffuser is initiated. For real systems the differentiation between both types of instabilities can not be made as sharp as suggested by the model. In an intermediate region a mixture between surge and stall occurs which is called classic surge.

Rotating stall

Rotating stall is understood as a travelling non-axisymmetric distortion of the circumferential flow pattern. There are many reports on the phenomena of rotating stall in axial and radial compressors in the open literature (e.g. Emmons et al. [23], Jansen [44], Senoo and Kinoshita [75, 77], Hunziker and Gyarmathy [37] or Spakovszky [84]). The type and mode of rotating stall are specific in amplitude, frequency and coupling for each machine.

In compressors equipped with vaned diffusers typically a single cell rotating stall is found while in unvaned diffusers multi-cell stall patterns are reported. As rotating stall is associated with a movement of flow distortions around the diffuser a phase shift of pressure fluctuations can be found at different circumferential positions within the diffuser (Figure 3-5). The traveling speed of the flow distortion is a fraction of impeller rotation speed. Each stall cell rotates within the diffuser at a fraction of 20 to 70 % of the rotor speed.
(Greitzer [31]) but travelling speeds of only 5% rotor speeds have also been reported (Di Liberti [17]). Gyarmarthy et al. [33] report on measurements which show a dependence of the rotational velocity of each stall cell and the mass flow rate.

Rotating stall in vaned diffusers

Figure 3-6 illustrates the propagation of a fully developed single-cell rotating stall in a vaned diffuser. The data were obtained by Irmler [42] with a 2D Laser Doppler Anemometer system at mid-height of the diffuser. The arrows depict the flow velocities and its direction throughout the diffuser passage. The stall cell propagates in the direction of impeller rotation around the annulus of the diffuser. It covers nearly 40% of the diffuser channels. At the front of the stall cell (Region A) the velocities at the diffuser exit decrease, while in the throat the velocities are still pointing strongly forward. This leads then to a flow reversal in the mid of the diffuser channel and the stall cell is growing up. In Region B the stall cell is fully developed and the forward flow breaks down completely. In the fully developed stall cell (Region B) reversed flow is found on the pressure side of the diffuser vanes while zero velocity is measured in the throat. In Region D the stall cell is washed out by strong forward flow, the diffuser channel recovers and a strong forward flow is re-established.
3.1 Definition of the unsteady phenomena

Rotating stall in vaneless diffusers

In vaneless diffusers more complex stall patterns are established. A variety of authors reports on stall pattern with more than one stall cell which are all rotating at the same speed within the vaneless diffuser. Figure 3-7 shows a sketch of a 2-cell rotating stall. Each of these stall cells can be understood as a fluctuation of velocity and pressure. The stall cell is covering the entire area between diffuser inlet and outlet. Each of these regions is traveling with a fraction of the impeller speed. For multiple cell stall patterns, this could result in a pressure fluctuation with a higher frequency than the rotor speed.
Chen et al. [9] explain the multi-cell rotating stall as a pattern composed by a Rossby wave inside the impeller which is associated with a circular Karman vortex street and show an analogy to the meteorological wind pattern at medium latitudes. The Rossby wave is formed due to instabilities between reversed high entropy fluid and forward moving low entropy fluid under the effect of the Coriolis force. The Rossby wave and thus the rotating stall cell propagation is coupled to the impeller speed and the strength of the interaction between forward and reverse flow.

### Coupling of stall and surge

Neither rotating stall nor surge can be considered as a stationary condition. The compressor is able to switch within a few rotations into stall, surge or back to stable flow. It is extremely hard to predict the exact time of switching as it could be triggered by very small perturbations and appears chaotically. This switching behavior can be seen in Figure 3-8, where a short time fourier transformation (STFT) or spectrogram of a pressure signal obtained in a vaned diffuser at $\text{Mu} = 0.4$ is shown. A STFT is a windowed discrete-time Fourier transform of a signal using sliding windows where the frequency spectrum is only calculated for that small time window. The plot reports the time along the x-axis while the frequency content of a portion is plotted along the y-axis. Stall is seen as a distinct frequency band (Line C) at a fraction of 29% of the impeller rotational speed. Surge is seen as a band at low frequencies (Line D). At time A only surge but no stalling event was found while at time B fully developed rotating stall is present. This indicates a coupling of stall and surge which is referred to as “classic surge” (Rose et al. [70]).
Inlet tip recirculation

Based on measurements of the static wall pressure along the shroud wall Hunziker [36] postulates the existence of a stationary recirculation zone at the inlet of the impeller. Ribi [64] confirmed the existence of this zone in the “Rigi” facility by using a pneumatic probe measurement close to the impeller inlet. He showed that the onset of inlet tip recirculation is caused by incidence at the leading edge of the impeller and occurs below a certain volume flow condition if no other instability occurs first. Recently a variety of authors (Hunziker et al. [38], Ishida et al. [43] and Dickmann et al. [18]) has shown that the occurrence of inlet tip recirculation could be delayed by introducing a circular chamber at the inlet. This chamber acts as a recirculation area where fluid is transported from the impeller back to the compressor inlet.
3.2 Characterization of the operating condition

For a characterization of the behavior of the compressor system an automated method was developed (Figure 3-10). For both scales of compressor systems investigated within this work the characterization is based on the same approach of analyzing the frequency content of a time resolved pressure signal. Based on the dominating frequency of the pressure signal a decision is made whether the compressor system is stable or surged. If the system is within rotating stall additionally a path-time method is applied to define the number of stall cells and their speed of rotation.

The fundamental tool in the characterization of the operating condition is the Fast Fourier Transformation (FFT) of a time resolved pressure signal. The automated characterization of the compressor behavior is based upon the signal obtained from sensor P9. This sensor is located on a radius equivalent to 105% of the impeller exit radius on the diffuser front wall. The location of this sensor is shown in Figure 3-11.

Figure 3-10 Sequence of the automated characterization method
This sensor location was used because the pressure signal obtained at this location is not dominated by the blade loading but it is still close enough behind the impeller exit that the jet-wake pattern still persist. Using this sensor rotating stall and surge can be reliably detected. Instabilities occurring at the inlet of the impeller cannot be detected.

The waveform and frequency content of the acquired pressure signal changes as the compressor is throttled from stable conditions into stall and surge. Figure 3-12 gives examples of the pressure readings and the FFT of the signal at four operating conditions. Conditions (A, B & C) have been acquired at part speed ($\text{Mu} = 0.6$) while the condition (D) has been acquired near the design speed at $\text{Mu} = 1.26$. In the upper part of each sub-plot the pressure fluctuation at 105% impeller exit diameter and the once-per-revolution shaft trigger are shown. In the lower plot the FFT is shown. In the FFT plot the amplitude information is normalized by using the strongest amplitude found in the spectrum. For the stable conditions (A)-(B) the frequency spectrum is dominated by the blade passing frequencies. At the condition close to stall (B) some frequency content at lower frequencies is found but the signals reveal a stable condition. In rotating stall the low frequencies become dominant and the relative amplitude at a frequency related to the blade passing is below 10% of the relative amplitude at a frequency related to stalling phenomena. This is also seen in the raw signal where in stall the blade passing is superimposed by the low frequency stalling event at a much stronger magnitude. In the case of surge (D) the surge frequency becomes dominant. The frequency of surge only depends on the geometry of the compression system and is not affected by the impeller speed.
The major drawback of the FFT-method is that no information on the occurrence of a rotating pattern is gained and thus no clear distinction on the presence of rotating stall can be made. A judgment on the number of stall cells or on their direction of rotation cannot be made based on one FFT only. Furthermore the FFT methods are limited if system instabilities occur only for a short time or if the instability is superimposed by acoustical waves or mechanical vibrations. Therefore the exclusive use of a FFT method for the characterization of rotating stall event is insufficient. Therefore an additional path-time method is applied for the stalled case.

Figure 3-12 Example of the time resolved pressure reading in the diffuser at Stable (A), Near Stall (B), Rotating Stall (C) and Surge (D)
A variety of authors (Levy et al. [57]; Spakovszky [84]) used a graphical path-time approach to define the number of stall cells and their direction of propagation. The path-time approach was automated in order to facilitate judgement on a variety of measurement points within acceptable computation time. The methods is robust, and once the frequency of the stall is known, only a few stall cycles needs to be evaluated to identify the direction and speed of the stall pattern of subsequent samples. In this work an automated method used which is based on the detection of a phase shift due to the rotating pattern in more than one pressure signal. If a rotating pattern is present, two pressure sensors located at different angular positions will show a phase shift, which is represented by a time lag on the pressure signals.

Within the path-time analysis, band pass filtered pressure traces are plotted corresponding to their normalized circumferential position from bottom to top in the direction of impeller rotation. At a normalized angular position of 1, corresponding to 360 degrees, the pressure signal of the first sensor is repeated assuming the impeller has accomplished one revolution. An example for this graphical approach is given in Figure 3-13. The graph constructed in this way resembles a path-time diagram. Assuming the rotational speed of the stall cells remains constant, the maxima of the pressure signals corresponding to one particular stall cell can be connected by straight lines. The steep red line in the left part of the figure represents the rotational speed of the impeller.

![Path-time diagram during rotating stall](image)

**Figure 3-13**  Path-time diagram during rotating stall
For the case shown in Figure 3-13 a four-cell stall structure is present. This is indicated by the repeating pattern for the fourth period. It is seen that each stall cell travels at a much lower angular velocity than the impeller. For the present compressor a fraction of 24 % to 28 % of the rotational speed was calculated depending on the throttle condition.

In the actual compressor, the raw pressure signals typically do not have a clear sinusoidal pattern. Instead they are composed of a variety of frequencies. Usually frequencies related to the rotating stall, surge, blade passing and even mechanical vibrations and their sub harmonics are present in the raw-output of a sensor. Therefore the signals need to be conditioned by narrow-band filtering before a rotating event can be characterized in respect to its speed and direction.

3.3 Stability of the compressor stage

Figure 3-14 shows the operating map of the enlarged research facility at the design tip clearance ratio of CR = 12.7% (Case “STR 17.2”). In the figure the total-to-static pressure ratio is plotted over the mass flow at an inlet reference condition of 293.15 K and 98.1 kPa. The design pressure ratio of $\pi = 2.8$ at the design mass flow of $m = 4.1$ kg/s is matched at the design speed of $Mu = 1.33$ (22,000 RPM).

The symbols describe the operating conditions according to the definition in Section 3.1. Towards high mass flow conditions, the operation range is limited by the choking of the impeller throat. For part speed, the resistance of the piping system is setting the maximum mass flow. Towards low mass flow conditions, the range is limited by stall and surge. For high-speed conditions, surge of the system occurs first. In an intermediate region classic surge seems to be the dominant feature. In classic surge, the characteristic frequencies of stall and surge modes are coupled. Isolated rotating stall pattern could only be identified for part speed conditions where surge is not present.
In Figure 3-15 an enlarged portion of the behavior map is plotted for part speed conditions. Again, the stall and surge regions are marked. The different marker indicate the status of the system at each operation point. They are determined using the FFT and path-time methods described above. In this figure the number of stall cells during isolated rotating stall can be studied.
The analysis reveals that under isolated rotating stall conditions a four-cell forward rotating stall pattern is present in the vaneless diffuser. Very close to the surge margin, the stall pattern is changing and a three-cell pattern becomes dominant. All stall patterns are rotating in the direction of the impeller rotation. The traveling speed of the rotating stall pattern depends on the mass flow condition: close to the stall line a fraction of 24% rotational impeller speed was calculated. Towards the surge line the stall cells are accelerating and a traveling speed of 28% of the impeller speed was found. As the rotating pattern is determined by using band pass filtered data, only the first order of the stall waves can be evaluated. The existence of higher order stall waves as described by Spakovszky [84] could not be verified for this compressor. As the current set-up consists of a vaneless diffuser no counter rotating patterns could be present as they only occur if a vaned diffuser is located at a certain distance downstream of the impeller.

*Figure 3-15  Enlarged cutout of the stalling behavior for the large-scale facility “Rigi”*
In Figure 3-16 the operating conditions at part- and full-speed are shown in the terms of the non-dimensional specific work- and flow coefficient. The onset of instability is marked. For all three conditions, the instability occurs at a specific work coefficient of 0.52 to 0.53. For a higher impeller speed instability occurs at a higher specific flow rate than for part speed. This is well known in centrifugal compressor as the destabilizing effect of the upstream storage volume increases with higher pressure ratios.

**Comparison of the enlarged and the full-size compressor**

In Figure 3-17 the overall compressor characteristics are shown for the true-scale application “miniRigi” and the enlarged facility “Rigi”. The experiments were performed in both facilities at the same Reynolds number. To compare the operating conditions the mass flow is non-dimensionalized with the design conditions of the two facilities. The line on the left side of operational data describes the onset of instability.
For corresponding non-dimensional parameters and an identical relative tip clearance the same operational characteristic is obtained at all investigated impeller speeds. For both experiments stall occurs at the same normalized mass flow. In the unstable regime some system level variation is present. Towards higher mass flow differences in the system resistance were found due to differences in the attached piping and cooling devices but the same choke limit was found. The stall pattern was studied in detail by applying the path-time algorithm described in Section 3.2 in both facilities. For both machines a forward rotating 4-cell stall pattern was found for most operating conditions. In some cases close to the stability margin the actual scale compressor “miniRigi” shows a 5-cell pattern. The traveling speed of the stall pattern is in both cases around 24 to 28% of the impeller speed.

For the highest speed line a similar choking limit was found, as choke is only dependent on the impeller geometry and not on the attached subsystems. The surge margin was defined in order to quantify the stable operating range:

\[
SM = \frac{V_{Choke} - V_{Stall}}{V_{Choke}} \tag{3.3}
\]
The surge margin was found to be 37% for both compressor scales at design speed.

These investigations reveal no differences in the stalling behavior and range between the actual scale and the enlarged compressor system. A summary and comparison of the findings is given in Table 3-1 to 3-3.

**Table 3-1  Comparison of the compressor behavior at surge**

<table>
<thead>
<tr>
<th>SURGE @ Mu=1.33</th>
<th>miniRigi</th>
<th>Rigi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surge Frequency</td>
<td>[Hz]</td>
<td>12</td>
</tr>
</tbody>
</table>

**Table 3-2  Comparison of the compressor behavior at rotating stall**

<table>
<thead>
<tr>
<th>STALL @ Mu=0.6</th>
<th>miniRigi</th>
<th>Rigi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller Frequency</td>
<td>[Hz]</td>
<td>1000</td>
</tr>
<tr>
<td>Stall Frequency</td>
<td>[Hz]</td>
<td>1147</td>
</tr>
<tr>
<td>Relative Stall Speed</td>
<td></td>
<td>28.6%</td>
</tr>
<tr>
<td>Rotating Stall Pattern</td>
<td></td>
<td>4 (&amp;5) Cell</td>
</tr>
</tbody>
</table>

**Table 3-3  Comparison of the surge margin**

<table>
<thead>
<tr>
<th>SURGEMARGIN @ Mu=1.33</th>
<th>miniRigi</th>
<th>Rigi</th>
</tr>
</thead>
<tbody>
<tr>
<td>Volume Flow @ Surge</td>
<td>[m³/s]</td>
<td>0.066</td>
</tr>
<tr>
<td>Volume Flow @ Choke</td>
<td>[m³/s]</td>
<td>0.105</td>
</tr>
<tr>
<td>Surge Margin (Eqn 3.3)</td>
<td></td>
<td>37%</td>
</tr>
</tbody>
</table>
3.4 Influence of the Reynolds number

In Figure 3-18 the influence of the Reynolds number on the stability behavior at part speed is shown for the large-scale facility “Rigi”. The closed loop inlet pressure was modified from 300 mbar to 1000 mbar. Therefore, by changing the inlet density a variation of Reynolds number by a factor of 3.3 was covered in the experiment. For the low Reynolds numbers the operating map reveals a drop of 0.5% in the total-to-static pressure ratio. This drop of the pressure head is also seen in a drop in efficiency as predicted by the relation given by Casey et al. [7], due to increased losses in the viscous boundary layer.

Towards higher ranges of mass flow the operating range is shifted, as the system resistance line is affected by the change in Reynolds number. A change of the stall margin was not found. An influence on the choke point could not be quantified as choke could only be investigated at the design speed and no measurements could be taken at these conditions. It is concluded that for the range of Reynolds number investigated the stability of the compressor is not affected by viscous forces.

Figure 3-18 Reynolds number effects in the enlarged facility
3.5 Influence of tip clearance width on the performance

Measurements by Hong et al. [35] have shown for a different impeller a substantial influence of the relative tip clearance on the operational behavior. To improve the understanding of these effects, measurements with decreased tip clearance have been performed. In Figure 3-19 the comparison of the operating maps for clearance ratios of 12.7% and 4.5% show an improved characteristic for the reduced tip clearance case “STR 15.7”.

Figure 3-19 Operating map of the “Rigi” facility at design tip clearance and reduced tip clearance
The pressure ratio is improved by 6% for the same impeller speed. The choke and the system resistance limit remain unchanged. Towards lower mass flows the instability limit is not affected for high impeller speed while for low impeller speed an improvement is found for the small tip clearance case. At high impeller speed, the surge margin remains unchanged as surge depends mainly on the overall setup of the compression system and is not affected by changes of the local flow pattern at impeller exit due to tip gap variations.

In Figure 3-20 an enlarged section of the operational characteristic is shown for part speed operation. The gain in stability margin of the small clearance case is due to a shift of the onset of rotating stall towards lower mass flows. The shift of the instability line towards lower mass flow is also seen in Figure 3-21 in terms of the non-dimensional specific work- and flow coefficients. The reduction in pressure rise and a shift of the instability onset towards higher flow coefficients are seen for the large clearance case. Thus, the onset of rotating stall is affected by flow features which depend on the tip clearance. In the further work it will be shown by detailed measurements how the onset of rotating stall is affected by the tip clearance flow.

Figure 3-20 Enlarged operating map of the “Rigi” facility at design tip clearance and small tip clearance
Influence of diffuser width

In the comparisons shown above only variations of the relative clearance ratio have been shown and the effect of the diffuser width was neglected. Up to now in “Rigi” the influence of the changed diffuser height on the performance was unclear. Therefore a stepped diffuser was developed which accounts for the change in diffuser width if the tip gap is varied by applying a shim plate. The stepped diffuser has a similar tip clearance ratio of CR = 12.7% as the baseline case with large clearance “STR 17.2”. The diffuser height is varied to study diffuser width effects on the stability and compressor performance. This is of importance as the diffuser height was changed while comparing the baseline clearance and the reduced clearance case “STR 15.7” (see Figure 2-14).

In Figure 3-22 the characteristic of the stepped configuration “STEP 15.7” and the baseline case “STR 17.2” are plotted for matching impeller speed. The performance towards the choke line is slightly improved for the stepped diffuser. The diffuser width has no effect on the maximum pressure rise in the design point and the slope of the characteristic. In this machine he onset of instability is not affected by the change in the diffuser width.
This is in contrary to the traditional layout of vaneless diffuser where the diffuser width is one of the key parameters (Senoo et al. [75-77]). In the current case, the reduction in pressure rise reported above for small and large tip gap are due to flow effects and losses caused by the change in tip gap and do not depend on the diffuser width. The losses due to the tip gap flow cannot be compensated with a stepped diffuser. Therefore no further investigations have been carried out on the stepped diffuser.
Influence of a pinched diffuser

In industrial applications it is common to use pinched diffusers to improve the instability limit of centrifugal compressors. A pinched diffuser with a pinch angle of 5 degree and a diffuser exit width of $b_5 = 11.2$ mm was designed. The pinched diffuser is compared to the baseline diffuser configuration. For both diffuser designs a similar tip clearance ratio of $CR = 12.7\%$ was tested. In Figure 3-23 the operating characteristics of the pinched and the baseline configuration with parallel wall diffuser are plotted.

![Operating map of the “Rigi” facility at design tip clearance rate with and without pinched diffuser](image-url)
Due to the pinch the pressure recovery in the vaneless diffuser is diminished resulting in a reduced stage pressure ratio. The choke limit and the system resistance line remain unchanged. For the pinched diffuser the onset of surge is shifted towards lower mass flow rates. In part speed the onset of rotating stall was delayed even though a positive slope of the characteristic reveals a potentially unstable behavior. The positive slope is indicating, that the pinched diffuser is able to damp the dynamic unstable system and is stabilizing the overall compressor. But, due to the positive slope of the characteristic the observed shift of the surge line is of no technical interest. In practice compressors can not be operated in regimes where the slope of the operating line is positive. Therefore, no further investigations have been carried out on the pinched diffuser.

Selected diffuser configurations

The strongest influence on the stability and behavior was observed for variations in the relative tip clearance. No significant differences have been seen in the case where only the diffuser width has been changed. For the case of a pinched diffuser a benefit was seen in the system analysis as the stall line was shifted towards lower mass flows. But the observed enhancement of the stability might not be of technical interest as the positive slope of the compressor operating line indicates an unstable condition.

For further investigations on the stability and the flow pattern at impeller exit the configurations “STR 157” and “STR 172” are selected. For this geometries detailed measurements of the flow structure in the diffuser are performed.
4 Evolution of the Flow in the Vaneless Diffuser

In Section 3.5 a significant change in the pressure rise and range of the compressor was found for different relative tip clearance configurations. This chapter will give a detailed description of the principal flow pattern found in the vaneless diffuser of a highly loaded turbocharger compressor. The aims are to describe and analyze the major flow pattern at stable and near stall operating points. As shown previously, the operating behavior of the compressor stage is similar between the true-scale and the enlarged compressor. Thus, measurements within the large scale machine are representative of the flow structures in a small scale compressor.

As a baseline case for studying the evolution of secondary flow the configuration “STRAIGHT 17.2” with a parallel wall diffuser and a clearance ratio of CR = 12.7% has been investigated in detail. This geometry is representative for the compressor arrangement in automotive turbocharger. In order to identify the flow features temporally resolved LDA measurements have been performed at different locations inside the diffuser. The temporally resolved data has been obtained for an part speed operating condition Mu=0.6 for a flow coefficient of PHI = 0.0513. At this mass flow rate the design incidence of 2 degree is present at leading edge of the main blade. The investigated operating condition is marked in Figure 4-1 by a full round circle.

![Figure 4-1](image-url)  
*Figure 4-1 Operating condition for the time-resolved flow measurement*
By using a time resolved techniques like the LDA measuring system the pitch wise flow variation can be resolved. Based on the rotor position the circumferential flow structure within the diffuser is determined by a phase locked ensemble averaging approach. As no downstream disturbances are present the flow structure is assumed to be circumferentially periodic and the time axis and circumferential axis coincide. Using this assumption one yields for each axial traversing line a complete iso-radial plane of velocity data. In this plane the flow is resolved relative to the impeller position and the secondary flow structures can be studied. For the time-resolved measurement, a total of 26 x 180 x 29 phase lock averaged points of the LDA data have been acquired within the vaneless diffuser. This unique data set provides an insight into the development and migration of the 3-dimensional flow within the diffuser.

4.1 Circumferentially averaged diffuser flow pattern

In Figure 4-2 measurements of the circumferentially averaged radial and absolute tangential velocities are shown for the baseline clearance case at seven radial positions within the vaneless diffuser. The experiments have been performed at an operating condition with a corrected mass flow rate of 1.95 kg/s (PHI = 0.051) and design incidence at impeller leading edge. For this plot the data has been time averaged for each axial and radial location. The spatial resolution is 0.5 mm in hub-shroud direction throughout the diffuser width. Due to limitations of the LDA technique no data was obtained very close to the diffuser hub wall. The black dotted lines define the radial position, while the colored lines show the magnitude of the radial and the tangential velocity, respectively. For negligible tangential velocities at the inlet the mean tangential velocity component is proportional to the Euler work input. The component of the radial velocity is equivalent to the mass flow distribution throughout the diffuser channel.

Even at the stable operating condition shown, a blocked zone with zero radial velocity is present very close to the shroud wall. The zone of low radial velocity extends from the shroud wall up to one third of the channel width. Very close to the shroud wall reversed flow is encountered. The radial velocities at mid height and at the hub wall are positive and no flow reversal is observed at the diffuser hub wall. At diffuser exit no flow reversal is found across the whole span of the diffuser.
The absolute tangential velocity is high throughout the diffuser. The highest values are measured at mid span while towards the diffuser walls the velocities are slightly lower. The diffusion process in the vaneless diffuser is seen in smaller absolute tangential and radial velocities at diffuser exit compared to the values at the inlet. Towards the diffuser exit mixing is taking place and the radial velocity gradient becomes smaller. The low radial velocities near the shroud indicates a separated zone where high loss fluid due to tip clearance flows has accumulated. According to the system analysis presented in Section 3.3 the local flow reversal at the shroud does not seem to affect the stability of the compressor system. The system is stabilized by the mixing processes throughout the diffuser which is seen in the positive radial velocity at diffuser exit.

In Figure 4-3 the velocity triangles are shown for the design incidence operating condition ($PHI = 0.051$) at two axial locations near the hub and the shroud wall. The triangles have been calculated from the circumferentially averaged data obtained at axial locations of $z/b = 15\%$ and $85\%$, respectively. The velocity triangles visualize the strong hub-to-shroud variation of the flow approaching the diffuser. Remarkable is in both triangles the slip of the flow relative to the impeller. The slip is resulting in relative flow angles $\beta_2$ which are much larger than the exit angle $\chi_2 = 30^\circ$ of the blading. The unequal
velocity triangles indicate skewed flows near the shroud due to the influence of the large relative tip gap and the meridional curvature of the channel. The deficit in the radial velocity near the shroud indicates a lack of mass flow in the shroud region. Therefore, simplifying assumptions which neglect axial gradients at impeller exit may not be valid for the description and modelling of the flow entering the vaneless diffuser. A similar skew of the flow in the diffuser was also found to a lower extent in the tip region of an industrial impeller in a previous investigation performed in the “Rigi” facility (Schleer et al [72]).

Figure 4-3 Velocity triangles near the hub and shroud for the baseline operating condition at $Mu=0.6$ and $PHI=0.051$

Figure 4-4 shows a comparison by Roduner [65] on data provided in the open literature for other industrial designs. Plotted is the radial velocity distribution for different impeller designs. The velocity distribution obtained for the current configurations is added to the comparison using round symbols. For a better comparability the data are plotted in a non-dimensional manner using the maximum radial velocity value and the relative diffuser width.

A similar trend with the lowest radial velocities near the shroud is seen in all examples. But, unlike the other examples the flow deficit near the shroud is much more pronounced for the current case and is showing reversed flow near the shroud wall. This could be due to the increased clearance ratio for the current design but also due to differences in the relation of meridional and circumferential accelerations for the different designs.
4.2 3-dimensional flow features in the vaneless diffuser

The classical jet-wake flow model in the literature

The governing feature usually found at the exit of a centrifugal impeller is the so-called “Jet-Wake” flow pattern which was introduced by Dean and Senoo in 1960 [15]. Figure 4-5 shows the idealized jet-wake model of a two-dimensional flow without axial variation. In the 2-D Jet-Wake model it is assumed that most of the flow leaves the impeller in the jet region. This region has an appropriate stagnation pressure for a nearly loss-free flow in the relative frame of reference. The jet is found on the pressure side of the blade and shows the highest radial velocity.

However, a large region of the impeller exit can be covered by the wake region. The wake region is found on the suction side of the blade and shows a much lower radial velocity but a very high tangential velocity. The wake is explained as a region where flow which has experienced high loss has accumulated. The high loss zone grows throughout the impeller passage and is finally recognized at the impeller exit as a wake. Experimental evidence for this model was given by Fowler [27] who showed the existence of a non

Figure 4-4  Comparison of the radial velocity distribution at the exit of a centrifugal compressor (modified from Roduner [65])
uniform flow in a slowly rotating huge impeller. Eckardt [21 and 22] and Krain [53] measured the meridional velocity at several measurement planes inside the impeller and confirmed the model. Since then the two dimensional model was constantly improved (Rohne and Banzhaf [67], Japikse [46]) and has been widely used in a two-zone approach for the preliminary design of centrifugal compressors (Japikse [45]).

![Figure 4-5](image)

**Figure 4-5**  *Idealized Jet-Wake model (from Dean and Senoo [15])*

**Flow profiles in the given compressor**

For further interpretation of the stability behavior of the current machine time resolved measurements of the flow structure are performed in the vaneless diffuser. As baseline for the explanation of the complex 3-dimensional flow pattern the straight diffuser with a diffuser width of $b_2 = 17.2$ mm ($CR = 12.7\%$) is chosen as a baseline. All measurements are performed at an operating condition of $Mu = 0.6$ and $PHI = 0.0513$. Based on these measurements the 3-dimensional flow structure, mixing, and evolution of the secondary flows are studied in detail.

First, the flow structure is observed in a hub-to-shroud view. The findings of these investigation are summarized in Table 4-1. Later, the evolution of the flow pattern throughout the diffuser is studied based upon iso-radial plots. For comparison the effect of the clearance ratio on the flow pattern is also shown for the reduced clearance case. Based on these measurements the 3-dimensional evolution of diffuser flow and effects due to the blade loading and the influence of the tip clearance are studied in detail.
In Figures 4-6 contours of the radial and absolute tangential velocities are plotted for a mid-span cut. Near the suction side high radial are found. Directly at impeller exit the absolute tangential velocities at the suction side is high, while farther downstream a region of low tangential velocity is present. On the pressure side the radial velocity is low while the tangential velocity is high. This does not agree to the classical Jet-Wake model by Dean and Senoo [15]. According to their Jet-Wake flow model a jet appears on the pressure side as a region of high radial velocity and low tangential velocity at the pressure side. On the suction side a wake region is expected which shows lower radial and higher absolute tangential velocities than the jet region on the pressure side.

(A) Radial Velocity

(B) Absolute Tangential Velocity

Figure 4-6  Measured contours of the radial and absolute tangential velocities at mid span ($z/b_2 = 50\%$) for the baseline clearance ratio case $CR = 12.7\%$
In Figure 4-7 the circumferentially resolved pattern of the radial and tangential velocities is plotted for an axial location close to the hub wall. Close to the hub the blade-wake pattern can be identified as a region of lower absolute tangential velocity in the contour plot (Figure 4-7.b). The jet region is found close to the pressure side while the wake region has established as a region of high absolute tangential velocity on the suction side as expected in the classical 2D-Jet-Wake model.

**Figure 4-7**  Measured contours of the radial and absolute tangential velocities near the hub ($z/b_2 = 85\%$) for the baseline clearance ratio case $CR = 12.7\%$
The reason for the high radial velocity near the suction side becomes clear if the flow pattern is investigated near the shroud wall (Figure 4-8). As an additional pattern, tip leakage flow is seen in this figure. This tip leakage flow pattern is seen at the shroud wall as a zone of very low and in some locations even reversed radial velocity next to the suction side. The tangential velocities in that regions are low as well. Evidence for the presence and influence of the clearance flow was seen in the velocity triangles near the hub and shroud wall (Figure 4-3). The velocity triangles indicate skewed layers and showed non-uniformity in hub-to-shroud direction with low radial velocities on the shroud.
As in the region affected by the tip leakage flow the radial velocity becomes very small or even negative, this zone appears as blockage of the diffuser channel. Due to this blockage effect the streamlines of the wake are diverted and the flow is accelerated radially. Therefore the radial velocity in the wake region is higher than expected and the wake appears in the vicinity of the blocked zone as a region of high mass flow.

The observations on the inverted jet wake pattern and the assignment of the flow pattern locations are summarized in Table 4-1. From this comparison is concluded that for the machine and operating condition investigated the classical Jet-Wake model is not sufficient to describe the flow entering the diffuser. The reason for the observed disagreement is the hub-to-shroud variation and 3-dimensionality of the flow. The classical Jet-Wake flow model was proposed as a purely 2D-description of the flow. This assumption is not fulfilled for the present case as the flow is governed by the hub-to-shroud variation and tip clearance flow effects.

The effect of the mass flow rate on the flow pattern and the distribution of the flow features is elaborated in detail in Section 4.5. In this section an improved description which is taking the effect of tip leakage into consideration is proposed.

Table 4-1 Characteristics of the flow pattern in the hub-to-shroud view

<table>
<thead>
<tr>
<th></th>
<th>Radial Velocity</th>
<th>Tangential Velocity</th>
<th>Pattern</th>
</tr>
</thead>
<tbody>
<tr>
<td>Classical Jet-Wake model</td>
<td>SS</td>
<td>very low</td>
<td>high</td>
</tr>
<tr>
<td></td>
<td>PS</td>
<td>very high</td>
<td>low</td>
</tr>
<tr>
<td>Mid span</td>
<td>SS</td>
<td>high</td>
<td>high</td>
</tr>
<tr>
<td>(z/b = 50%)</td>
<td>PS</td>
<td>low</td>
<td>high</td>
</tr>
<tr>
<td>Near the hub</td>
<td>SS</td>
<td>very high</td>
<td>high</td>
</tr>
<tr>
<td>(z/b = 85%)</td>
<td>PS</td>
<td>high</td>
<td>low</td>
</tr>
<tr>
<td>Blade</td>
<td>very low</td>
<td>high</td>
<td>high</td>
</tr>
<tr>
<td>Near the shroud</td>
<td>SS</td>
<td>reversed</td>
<td>low</td>
</tr>
<tr>
<td>(z/b = 9%)</td>
<td>PS</td>
<td>very low</td>
<td>high</td>
</tr>
</tbody>
</table>
Evolution and mixing of the flow pattern

Next, the interaction of flow structure which leads to the disagreement with the classical Jet-Wake model is studied. Figure 4-9 shows a 2D iso-radial view of the absolute tangential and radial velocities at a radial location of $R/R_2 = 104.25\%$. Figure 4-10 shows the same velocity components at a radial location of $R/R_2 = 116.25\%$. The constant radius plots reveal a strongly non-uniform flow in axial and circumferential directions. The main flow features are marked in the figures. The positions of the blades are marked according to the signal of the shaft encoder. As the velocity contours are plotted 4.25\% and 16.25\% impeller radii downstream in the diffuser the flow features are shifted relative to the fixed position of the blade.

In both figures complex flow features which cannot be explained by the simple 2-zone Jet-Wake model appear. The flow structure is highly three dimensional. In addition to the circumferential variation due to the blade loading the flow is affected in the axial direction by a skewed shear layer. In these time resolved representations additional 3-dimensional flow patterns are newly seen.

(A) Radial Velocity

(B) Absolute Tangential Velocity

Figure 4-9  Measured contours of radial and absolute tangential velocity at a location $R/R_2=104\%$ for the baseline clearance $CR =12.7\%$
The flow is not following one single log-spiral as a quasi-steady rigid body as it is assumed in the 2D models. Instead, the flow is following different trajectories for each axial and circumferential location. The flow patterns interact with each other and are able to change their location as they pass through the diffuser. In the following, the 3-dimensional secondary flow patterns are discussed and their trajectories within the diffuser are analyzed.

In Figure 4-9 the channel wake is seen as a region of high tangential velocities at mid-span and near the suction side of the blades. It is the strongest in the middle of the channel and is interacting with the clearance flow. Farther downstream the wake area has started to mix out but is still visible as a region of high tangential velocity. The jet appears as a region of low tangential velocity in the pressure side hub corner. The dominating pattern seen in the radial velocity distribution is the mass flow deficit towards the shroud. Due to the hub-to-shroud skew a clear distinction of the jet-wake pattern based on the radial velocities (Figure 4-9.a) cannot be made. This is not found in the 2D prediction by Dean and Senoo where the jet is described as the region with the highest radial velocity. In Figure 4-10 the jet has mixed out completely.

**Figure 4-10** Measured contours of radial and absolute tangential velocity at a location $R/R_2=116\%$ for the baseline clearance $CR=12.7\%$
The third flow pattern in Figure 4-9 is the “blade wake”. This small region next to the blades shows a low outward radial velocity and a high tangential velocity. It has the attributes of a wake but it appears directly at the trailing edge of the blade. As the jet region has already experienced a slip relative to the blade position while the trailing edge wake experiences only little slip, the blade wake and the jet interact strongly and interchange momentum. As seen in Figure 4-7 the blade wake mixes out rapidly with the jet flow. At a radius of $R/R2 = 110\%$ the blade wake has mixed completely with the jet flow. For comparison, the channel wake region is diffusing, but still visible throughout the diffuser until the last measurement location 125% downstream of the impeller exit.

The low radial velocity zone near the shroud is caused by the flow through the tip clearance. The tip clearance flow is seen at both radii as a layer of reversed flow near the shroud wall. In the clearance zone flow the lowest tangential velocity and a large slip velocity is found. In the contour plots the strongest tip clearance flow pattern occurs behind the tip of the splitter blade. At the full blade, the reversed flow is also present but not as pronounced. Farther downstream the clearance flow remains dominant (Figure 4-10). It migrates from the shroud wall towards the mid height of the diffuser. During its migration the flow mixes with the channel flow and flow reversal is no longer observed. This clearance flow region is identified in the stability assessment as a region of high flow distortion and turbulence (Figure 5-10). Based upon large pressure deviations a vortical structure which is interacting with the channel flow was postulated. In the same study also the differences in the intensity of the clearance flow at the main and splitter blade channels where postulated.

The tip leakage flow results in a blockage of the diffuser channel. Due to this blockage effect the streamlines of the wake are diverted towards the hub and the flow is accelerated radially. Therefore the radial velocity in the wake region is higher than expected. The wake appears in the vicinity of the blocked zone as a region of high mass flow. No significant difference of this flow mechanism is seen between main and splitter blades.
Randomness of the flow in the diffuser

In Figure 4-11 the randomness or deviation of the flow is given in terms of a turbulence level. The local turbulence level $Tu$ is defined as:

$$
Tu_{(r,z)} = \sqrt{\frac{1}{N} \sum_{i=1}^{N} \left( \frac{v_{i(r,z)} - v(r,z)}{v(r,z)} \right)^2}
$$

where the square root describes the deviation of the local velocity readings obtained by the LDA system which are normalized by the mean absolute value of the local velocity vector.

In the tip clearance flow zone very high turbulence levels of above 25% are measured while the lowest turbulence levels are recorded in a region at the suction side of the blades and in the pressure side hub corner. In the blade wake also elevated turbulence intensities are measured. This supports the existence of a vortical structure in the tip gap region and a strong interaction with the channel flow. The interaction mechanism of the tip clearance flow and the channel flow is different for the main and splitter blade channel. In the case of the splitter blade the interaction is strongest as the clearance jet is interacting with the rotational channel flow while at the main blade the leakage interacts with the non-rotational fluid entering the impeller. The interaction of the tip gap and the channel flow is also seen in the wall pressure fluctuation shown in the stability assessment (Chapter 5). A similar conclusion on the existence of vortices can be drawn for the blade trailing edge region, where turbulent mixing of the blade wake and the jet is seen.

Figure 4-11 Turbulence level at a radial location $R/R2=104\%$
Velocity triangles for the secondary flow pattern

The velocity triangles for the main zones at impeller exit are plotted and labeled in Figure 4-12. The circumferential and axial positions of the specific velocity triangles are marked in Figure 4-9 by small circles. In the velocity triangles the classical distinction between the jet and the channel wake pattern is found. The wake shows the highest absolute tangential velocity, while the relative tangential velocity is higher in the jet region.

![Figure 4-12 Velocity triangles for the dominating flow pattern](image)

No significant difference is found in the relative flow angle of the channel wake and the jet flow pattern. The wake near the blade shows a similar tangential velocity as the channel wake pattern but the radial velocity is substantially elevated due to the blockage effect of the tip clearance vortex. In the blade wake, a lower slip velocity is found than in the channel wake or in the jet pattern as in this region the absolute tangential velocity is high.

The clearance flow can be identified very clearly as an additional zone in the velocity triangles. In the tip clearance region the radial and tangential velocities are low while the relative tangential velocity and the slip velocity becomes large. The flow is not following the blade direction but is crossing the blade through the tip gap and appears as a zone of high blockage within the diffuser channel.
4.3 Vortical structure of the flow in the tip gap region

In Figure 4-13 velocity vectors are shown in a iso-Theta plane. The vectors are the radial and axial velocity component in the absolute frame of reference and the tangential velocity has been set to zero for plotting. Due to limitations in the optical access no data could have been obtained at impeller exit near the hub wall. The vectors are plotted for different circumferential positions relative to the splitter blade trailing edge. The position relative to the blade where the vectors are acquired is illustrated in the upper left corner of the plot.

Similar findings are seen behind the trailing edges of main and splitter blade and thus the discussion is restricted herein to the splitter blade only. In sub-plot (A) the vectors are shown for a circumferential position next to the pressure side of the splitter blade. A deficit in radial velocity near the shroud is observed. The deficit was also seen in the circumferentially averaged plot (Figure 4-4) and the time resolved contours (Figure 4-9). Towards the hub wall forward flow is seen. Very close to the tip region of the impeller exit the flow vectors point towards the shroud wall. This is indicating a filling up of the low momentum zone near the shroud by a movement of the channel flow towards the shroud wall. Farther downstream a blocked zone is seen near the diffuser front wall. This zone is a vestige of the previous tip vortex which is slowly convecting downstream towards diffuser exit.

On the pressure side of the approaching blade (Sub-plot (B)) a sudden change in flow direction takes place and the flow in the tip region is diverted towards the hub. This sudden change in flow direction is due to the influence of the leakage flow from the pressure to the suction side around the impeller trailing edge. In sub-plot (C) which was acquired at the suction side of the blade the flow at the trailing edge tip is pointing again towards the shroud while the radial velocity breaks down. This sudden change in flow direction and the local reversal of the radial velocity is indicating a vortical structure which is shed at the trailing edge of the blade tip. After leaving the impeller the tip vortex it bent backwards and as can be observed in the subsequent plots.
In sub-plots (D-G) reversed radial velocities are recorded in the vicinity of the shroud wall. In the vicinity of the reversed zone a blocked zone is forming where both radial and axial velocity become zero. This is indicating an intrinsically stable vortical structure which is convected with the flow towards diffuser exit. The vortical zone is migrating towards the diffuser exit and is shifted slightly towards mid-channel but remains visible in all plots.
The tip vortex migrates slowly towards the diffuser exit and blocks the diffuser channel. The channel flow is diverted towards the hub which results in a wavy flow pattern (Sub-plot F&G). Downstream of the blocked zone the flow is able to re-attach to the wall and positive radial velocities are recorded. Therefore the flow reversal observed in the vicinity of the tip vortex can be regarded as a local effect. The flow reversal occurs at one side of the vortex core and convects with the flow towards the exit of the diffuser. In regions not affected by the tip vortex the flow is attached and a stable forward flow is established. The local flow reversal does not cause a general flow reversal as it is observed during stall.

**Development of secondary flow pattern throughout the vaneless diffuser**

The plots of the velocity contours and the flow vectors do not allow a comprehensive assessment of the interactions between the secondary flow pattern. In order to understand the behavior and evolution of the flow pattern, perturbation vectors are introduced. The perturbation vectors allow a distinction of local deviations from the average. The vectors are defined as the difference of the local velocity and the circumferentially averaged velocity:

$$\tilde{v}(r, \theta, z) = v(r, \theta, z) - \overline{v}(r, z) \quad (4.2)$$

In Figure 4-14 the flow perturbation vectors are shown for four iso-radial planes. As in the previous representations the relative position of the blades are marked and labeled and the 3-dimensionality of the flow is observed. In both blade channels similar secondary flow patterns are found but a stronger tip vortex appears behind the splitter blade. This agrees with the finding from the distributions of the static casing pressure fluctuation given in Chapter 5, where more severe interactions of the tip vortex and the channel flows were reported at the location of the splitter blade.
Near the impeller exit (R/R2 = 102% and 104.25%) a movement of the flow along the blades from hub to shroud is observed. This movement in the axial direction results from a skew of the viscous layer fluid along the blades. The skew along the blades was seen in the velocity contours as blade wake. The mixing zone between Jet and the Wake fluid is seen as a sheared layer next to the pressure side. At the radial location R/R2 = 104% a corner vortex is found in both blade channels near the suction side hub corner. The flow perturbation vectors reveal a rapid mixing of the corner vortex and it cannot be identified at the location R/R2 = 109.5%.

At the suction side a strong shroud-to-hub movement and flow perturbation is observed. The perturbation is related to the tip clearance vortex and is seen at all radii as a zone of strong relative movement. The clearance flow is forming...
a vortex at the blade suction side tip and the trailing edge. As soon as the vortex enters the vaneless diffuser it is bent towards the suction side. In the diffuser the clearance vortex is migrating towards mid height of the channel and is causing the severe flow interactions. Beside the clearance vortex a large rotating pattern is found. This rotational structure is related to a passage vortex which is forming at the position of the channel wake.

4.4 Improved model for the description of the diffuser flow

The measurements on the flow structure in the vaneless diffuser revealed a significant non-uniformity of the flow at the discharge of the impeller. The flow entering the diffuser is severely skewed in axial direction. Near the hub high radial velocities are present while near the shroud the flow is locally reversed. Due to the high blade loading pitch wise flow variations are present. The jet and wake regions appear contradict to the predictions of the 2D Jet-Wake model by Dean and Senoo. A strong tip clearance flow pattern was identified near the suction side shroud corner as an additional flow feature. In the clearance zone only little work is done to the fluid and low absolute tangential velocities are found. The clearance flow is highly vortical and turbulent. It appears to the channel flow as a zone of high blockage. An analysis of the secondary flow pattern revealed a strong interaction between the tip gap zone and the channel wake.

Farther downstream in the diffuser the jet structure has mixed with the blade wake. The mixing process of the flow pattern takes place at two different time scales: The jet and the blade wake are mixing out rapidly at the impeller exit while the channel wake and the clearance flow remain stable throughout the diffuser.

In Figure 4-15 a schematic description of the secondary flow pattern is provided for two radial locations. This model applies to the description of a highly loaded centrifugal compressor with large relative tip clearance. It includes the strong effect of the low work leakage flow on the flow pattern and the different mixing scales for the blade wake and the clearance zones. The secondary flow patterns behave similarly to the predictions of the model proposed by Eckardt [22] for an impeller without backsweep.
4.4 Improved model for the description of the diffuser flow

The area covered by the tip clearance flow is larger than that in the Eckardt model due to the higher backsweep of the blading in the current study. The circular movement of the channel wake and the skewed layers along the blades are in good agreement with this flow model. Near the impeller exit the classical jet-wake structure and superimposed the effects of the tip clearance flow are seen as dominant flow features. The measurements reveal highly turbulent and vortical tip clearance flows.

The tip clearance flow has a strong vortical structure and is migrating towards the hub. Along the blades a skew of the viscous layers towards the shroud and a blade wake is seen. Circular flow perturbation vectors are found in the wake region and are causing a flow across the channel. The jet and the blade wake pattern are interacting rapidly near the exit of the impeller and a mixed zone is

---

**Figure 4-15**  
_Schematic model of the secondary flow pattern at impeller exit and at 120% impeller radius in the vaneless diffuser_
established. Therefore the Jet and Wake pattern cannot be identified farther downstream in the vaneless diffuser. The clearance flow region and the channel wake remain stable and can be seen up to the diffuser exit.

4.5 Diffuser flow pattern at off-design flow rates

The flow structure inside the vaneless diffuser was measured at different mass flow rates in two tip clearance configurations with the time resolved probe system (FRAP). The FRAP technology allows a simpler and less time consuming measurement of the three-dimensional flow within the diffuser than the optical LDA system. The operating points where measurements have been taken are marked in Figure 4-16.

![Figure 4-16 Operating conditions for time resolved FRAP measurements](image)

To cope with the expected differences in the yaw angle distribution at different mass flow conditions the FRAP probe was traversed using the intelligent self-aligning mode described in Section 2.10. The probe was installed at the radial location $R/R_2 = 105\%$ for all FRAP measurements. This measurement location corresponds to the location P9 (see Figure 2-20) which was used in Section 3.2 to assess the system stability. The same location was covered by the LDA measurements as discussed above. For validation of the experimental results a comparison of both measurement technologies is performed herein. The focus of the discussion is the influence of the mass flow condition and the diffuser configuration on the flow pattern entering the diffuser. First, the
influence of the mass flow condition on the flow structure will be studied for the large clearance case “STR 17.2”. The operating conditions where time resolved data were acquired are shown in Figure 4-1. In a later section the discussion will be extended to the small relative tip clearance configuration “STR 15.7” and comparisons at different flow rates are shown.

Time averaged velocities at off-design conditions

In Figure 4-17 the distribution of the time-averaged absolute tangential and radial velocity distributions across the diffuser width is shown for different mass flow conditions. On a second axis the flow angle in the absolute frame is plotted. All mass flow conditions shown are in the stable operating regime of the compressor as can be seen in Figure 4-1.

(A) PHI= 0.062
Near System Resistance Line

(C) PHI= 0.051
Design Incidence

(E) PHI= 0.035

(F) PHI= 0.022
Stall Line

Figure 4-17  Time-averaged distribution of radial and tangential velocities (FRAP) at a position of R/R2 = 105% for different mass flow rates (STR 17.2)
As the work input for the low mass-flow case is higher elevated levels of the tangential velocity are seen for the low mass flow condition. Near the shroud wall a deficit in the tangential velocity is seen for all conditions. This deficit of the absolute tangential velocity is equivalent to a low Euler work input or high losses in that particular zone. In stable, high flow rate conditions (Sub-figures (A&C)) a strong skew of the flow in the vaneless diffuser is seen in the absolute flow angle distribution.

Towards lower mass flow conditions the skew is reduced while at near stall condition no skew is observed and the flow appears like a full turbulent channel flow. At all stable mass flow conditions a local flow reversal occurs near the diffuser front wall. Close to the instability limit the present measurements indicate a recovery of the forward flow. Despite this flow reversal the system analysis shown in Section 3.3 indicated a stable behavior of the compressor system for the mass flow conditions (A) - (E). This is in contrary to the description in the literature (e.g. Senoo and Kinoshita [77]) where local flow reversal is linked to instability. An explanation for this behavior was given in Chapter 4.2 where the local flow reversal is explained as a localized flow reversal as a result of the tip vortex. As the tip vortex is pulsating with blade passing the flow is able to reattach and the reversed flow zone remains stable and is convected downstream. Therefore, even if a local flow reversal is observed no generalized backflow is occurring and rotating stall in the diffuser is not found.

For the condition near stall (Sub-figure (F)) the skew is not observed and a flat profile of the radial velocity is obtained. This indicates a high axial mixing of the flow near the stall limit. The increased mixing is associated with the occurrence of inlet tip recirculation postulated for mass flow conditions close to the stall limit. This trend is also seen in the static wall pressure fluctuation shown in Section 5.3. The strong axial mixing seen in the distribution of absolute flow angle. The mixing is preventing the local flow reversal in the vicinity of the shroud due to the tip gap vortex as it is seen in the stable conditions. Even though no reversed flow is found for the near stall condition the averaged flow angle is very shallow. Therefore rotating stall within the diffuser can be triggered if the mass flow is further reduced as a result of generalized flow reversal.
4.5 Diffuser flow pattern at off-design flow rates

Time resolved flow pattern at off design conditions

In Figures 4-18 and 4-19 contours of radial and absolute tangential velocities are given. Figure 4-20 shows contours of total pressure, which was not available for the LDA measurements discussed above. The total pressure is normalized by the stage inlet total pressure. In Figure 4-21 the absolute flow angle is plotted. For each variable the upper plot (A) was obtained at a mass flow condition close to the system resistance line while sub-plot (F) was acquired close to the stall limit. All plots are given using the same color-scale and the positions of the blades are indicated and labeled.

Due to the higher pressure ratio and therefore a higher Euler work input in the Cases (D) & (E) the tangential velocity is elevated while the radial velocity is lower due to the lower mass flow at these conditions. The skew of the flow is more pronounced for high mass flow conditions than towards the stall line. For high mass flow conditions (A&B) the classical Jet-Wake pattern appears inverted. A very small Jet region is found as a pattern of low tangential velocity and low total pressure in the pressure side hub corner. The wake region appears as a region of high tangential velocity in the middle of the passage but is shifted by clearance flow towards the adjacent pressure side. At the shroud the tip clearance flow establishes as a strong feature. The clearance flow shows low radial and tangential velocities. In the clearance flow area low total pressure values are found which is indicating high losses. For the mass flow condition near the stall line (F) a block profile without skew and very little circumferential variation is observed in both velocity components.

In Figure 4-21 a change in the absolute flow angle with mass flow rate is seen. As the mass flow rate is reduced, the flow angles become flatter but the hub to shroud variation is diminished. In the operating condition near stall (E) no hub-to-shroud variation is seen and the flow is leaving the impeller nearly tangentially. If the flow is throttled further, the flow is reversed and stall is initiated. The redistribution of the flow pattern towards lower mass flow rates is also seen in Figure 4-22 where the local angular momentum flux is plotted. The angular momentum flux is defined as:

\[
\hat{\rho}(r, \theta) = \dot{m}(r, \theta) \cdot \nu_{\theta}(r, \theta)
\]  

(4.3)

using the measured local velocity components the equation is re-written as:

\[
\hat{\rho}(r, \theta) = \rho(r, \theta) \cdot \nu_{r}(r, \theta) \cdot A_{r}(r, \theta) \cdot \nu_{\theta}(r, \theta)
\]  

(4.4)
All velocity components are measured in a time-resolved manner by the FRAP system while the density $\rho$ is known from the averaged probe head temperature and the time-resolved static pressure distribution. In hub-to-shroud direction the plots of the momentum flux reveal a redistribution of the work done to the fluid. In circumferential directions little variations are seen. If the mass flow is throttled, the pressure ratio gets larger and the level of the angular momentum is increased. At high mass flows the effect of the tip clearances flow is seen as a region of low angular momentum near the shroud. For near stall condition the local angular momentum is distributed uniformly across the channel.

In the Sub-plot (F) of Figures 4-18, 4-19 and 4-21 the behavior of the flow at near stall condition is seen. The flow appears as a full profile with little variation in the axial direction. No clearance vortex can be identified at near stall conditions. As the flow is still transported forward the system remains stable and pressure is built up. The uniform flow distribution suggest an increased mixing in the impeller passage. The blades are no longer loaded by the lift of the airfoils and the flow is not able to follow the airfoil but is pumped forward as a result of centrifugal force. This leads to the changed flow structure at the diffuser inlet and the flow pattern appears as a uniform wake.

In the circumferential direction a weak flow pattern which is similar to the classical Jet-Wake structure is formed. The velocity contours reveal higher radial velocities on the pressure side than on the suction side but the differences are weak compared to the gradients found at high mass flow rates. This structure is a consequence of the Coriolis force in the rectangular rotating channel and is seen as secondary flow structure at the diffuser inlet. This weak Jet-Wake pattern is also seen in the distribution of the angular momentum flux (Figure 4-22.F).

This change of the flow structure is occurring at the operating condition (F) where the stability analysis revealed a inlet tip recirculation. At the same operating condition the analysis of the static wall pressure fluctuation revealed a trajectory of the tip vortex perpendicular to the axis of the machine (Chapter 5). Based on these observations the flow interaction in the impeller can be postulated. As soon as inlet tip recirculation is triggered the loading in the inducer breaks down. This leads to an increased mixing of the flow and prevents the separation into jet and wake fluid. This hypothesis is supported by the observation in Figure 5-7.G where an unloading of the (axial) inducer is seen. Farther downstream in the radial section the fluid is still transported forward by the centrifugal effect and pressure is built up.
4.5 Diffuser flow pattern at off-design flow rates

(A) PHI= 0.062; Near System Resistance Line

(B) PHI= 0.055

(C) PHI= 0.051; Design Incidence

(D) PHI= 0.042

(E) PHI= 0.035

(F) PHI= 0.022; Stall Line

Figure 4-18  Radial velocity at R/R2 = 105% for the baseline tip gap configuration “STR 17.2” at different flow coefficients
4 Evolution of the Flow in the Vaneless Diffuser

(A) PHI = 0.062; Near System Resistance Line

(B) PHI = 0.055

(C) PHI = 0.051; Design Incidence

(D) PHI = 0.042

(E) PHI = 0.035

(F) PHI = 0.022; Stall Line

Figure 4-19 Absolute tangential velocity at R/R2 = 105% for the baseline tip gap configuration “STR 17.2” at different flow coefficients
4.5 Diffuser flow pattern at off-design flow rates

(A) PHI= 0.062; Near System Resistance Line

(B) PHI= 0.055

(C) PHI= 0.051; Design Incidence

(D) PHI= 0.042

(E) PHI= 0.035

(F) PHI= 0.022; Stall Line

Figure 4-20  Normalized total pressure at R/R2 = 105% for the baseline tip gap configuration “STR 17.2” at different flow coefficients
Figure 4-21  Absolute flow angle ALPHA at R/R2 = 105% for the baseline tip gap configuration “STR 17.2” at different flow coefficients.
Figure 4-22  Angular Momentum Flux at R/R2 = 105% for the baseline configuration “STR 17.2” at different flow coefficients

(A) PHI= 0.062; Near System Resistance Line

(B) PHI= 0.055

(C) PHI= 0.051; Design Incidence

(D) PHI= 0.042

(E) PHI= 0.035

(F) PHI= 0.022; Stall Line
Behavior of the tip clearance vortex

In the plots of the absolute tangential velocity at high mass flow rates (Figure 4-19.A-C) the tip clearance flow can be identified as a region of low absolute tangential velocity near the suction side shroud corner. A vortex core is seen as a region of very large gradients in the distribution of the radial and tangential velocities (Figures 4-18 and 4-19). These large gradients are also seen in the absolute flow angle plots (Figure 4-21). The strongest vortex appears in the case of the highest mass flow (Case A) and is stronger at the suction side of the splitter blade than at the main blade. For high mass flow conditions the tip vortex is able to migrate towards the middle of the channel while at lower mass flow the vortex remains near the shroud. This agrees with the observations shown in Chapter 5 based on static wall pressure fluctuations. There, the existence of a strong tip clearance vortex was concluded from the deviation of the static wall pressure and a stronger tip vortex was found for the splitter blade.

Secondary flow vectors

In Figure 4-23 the secondary flow structure is shown. The flow perturbation vectors are defined according to the definition given in Equation 4.2. In comparison with Figure 4-14 the flow perturbations appear less pronounced than in the LDA data where more details of the flow structure were resolved.

The position of the tip vortex is marked in Figure 4-23.A. The main features of the tip gap vortex are seen as a region of recirculating flow near the shroud and as a body of clearance fluid migrating towards the middle of the channel. These secondary flows occur due to the entrainment of the tip flow into the channel flow. In this region of recirculating flows the existence of a tip vortex was postulated based on gradients in the tangential and radial velocities. Additionally secondary flow structures are seen along the blade towards the blade tip on the pressure and suction side of the blades. The intensity of the secondary flow perturbation diminishes if the mass flow is throttled. For medium mass flow conditions (\(\text{PHI}=0.35\)) a very weak influence of the tip gap but no vortex core can be seen at all. At near stall condition (Figure 4-23.F) no influence of the clearance flow is seen. This is again supporting the assumption that the aerodynamic loading breaks down and increased mixing is taking place as a consequence of inlet recirculation.
Figure 4-23 Secondary flow structure at R/R2 = 105% for the baseline tip gap configuration “STR 17.2” at different flow coefficients
4.6 Explanation for the changed flow structure at off-design flow conditions

In the measurements it is seen that the position of the deficit area at the exit of the diffuser changes with the mass flow condition. This behavior was also observed before by a variety of authors. Rohne and Banzhaaf [67] presented measurements at the exit of a backswept impeller and reported a displacement of the wake area towards the shroud. They concluded that the classical jet-wake theory is not universally valid and needs improvement for backswept impeller. Johnson and Moore [47&48] and Inoue and Cumpsty [40] concluded that the position of the wake is governed by secondary flows due to curvature as well as due to rotational effects. They showed that for purely radial blading the position of the wake is controlled by the Rossby number, which is the ratio of inertia of a particle in the relative frame to the centrifugal Coriolis force:

\[ \text{Ro} = \frac{w}{\Omega \cdot R_c} \]  

(4.5)

For expressing the acceleration forces Cumpsty [12] used the radius of the curvature of a streamline in the relative frame \( R_c \) and the relative frame mean velocity \( w \). For purely radial blading the radius of curvature \( R_c \) is similar to the radius of curvature of the shroud in the meridional plane \( R_m \). For non-radial blading the relation of the forces due to streamline curvature and the centrifugal or Coriolis forces cannot be explained by the Rossby number alone as the blade forces act both in meridional and in circumferential direction.

Gyarmathy et al. [33] extended the description of the secondary flow development for non-radial blading by using the ratio of accelerations in the meridional and circumferential directions. They explained the displacement of the wake as a result of the changed forces on the flow at different mass flow conditions. To account for the secondary flows Gyarmathy et al. accounted for these effects by calculating the ratio of the resultant blade-to-blade acceleration \( \hat{a}_q \) and the resultant casing-to-hub acceleration \( \hat{a}_m \). This model is understood as an extension to the Rossby number for the explanation of the flow pattern at impeller exit which is valid in non-radial blading.

In the model of Gyarmathy et al. it is assumed that the fluid is travelling at a mean relative velocity \( \bar{w} \) and is following the geometry of the impeller. The direction and nomenclature of the acceleration components and their directions are indicated in Figure 4-24.
4.6 Explanation for the changed flow structure at off-design flow conditions

From the geometry of the impeller the relative velocities in the meridional and axial plane are derived as:

\[ \overrightarrow{w}_m = \overrightarrow{w} \cdot \cos(\beta) \]  \hspace{1cm} (4.6)

and:

\[ \overrightarrow{w}_a = \overrightarrow{w} \cdot \sin(\varepsilon) \]  \hspace{1cm} (4.7)

Forces due to the curvature of the streamline in the relative frame can be expressed by the relative velocity component perpendicular the radius of curvature. By doing so the accelerations in the meridional and circumferential directions are obtained as:

\[ \overrightarrow{a}_{Km} = \frac{\overrightarrow{w}_m^2}{R_m} = \frac{(\overrightarrow{w} \cdot \cos(\beta))^2}{R_m} \]  \hspace{1cm} (4.8)

and:

\[ \overrightarrow{a}_{Kq} = \frac{\overrightarrow{w}_a^2}{R_q} = \frac{(\overrightarrow{w} \cdot \sin(\varepsilon))^2}{R_q} \]  \hspace{1cm} (4.9)
Additionally to the curvature terms, a fluid particle is accelerated in the circumferential direction due to the rotation of the impeller:

$$\hat{a}_c = -2 \cdot \Omega \cdot \bar{w} \quad (4.10)$$

and in the radial direction:

$$\hat{a}_r = R \cdot \Omega^2 \quad (4.11)$$

The acceleration of a fluid particle along the streamline act perpendicular to the directions of interest. Therefore they do not need to be known for this description of the flow.

Each of the acceleration vectors defined in Equations 4.8 to 4.11 can have components in meridional as well as in circumferential directions which have to be taken into account for the resultant accelerations. For a given impeller geometry accelerations in the meridional and circumferential direction can be computed as:

$$\hat{a}_m = \hat{a}_{Km} - \cos(\varepsilon) \cdot (\hat{a}_z + \hat{a}_c + \hat{a}_{Kq}) \quad (4.12)$$

$$\hat{a}_b = \hat{a}_{Kq} + \hat{a}_c + \sin(\beta) \cdot (\hat{a}_z + \hat{a}_{Km}) \quad (4.13)$$

For purely radial impellers the relation of both accelerations is equivalent to the Rossby number as the streamline at the impeller exit is inclined by the angles $\beta = 0^\circ \rightarrow R_{Kq} \sim \infty$ and $\varepsilon = 90^\circ$. Using these geometrical values a linear relation between Rossby number and the acceleration ratio is found:

$$\frac{\hat{a}_m}{\hat{a}_q} = - \frac{\bar{w}}{2 \cdot \Omega \cdot R_{Km}} = - \frac{1}{2} \cdot \text{Ro} \quad (4.14)$$

Qualitative description of the influence of the transverse accelerations

The influence of these transverse accelerations $\hat{a}_m$ and $\hat{a}_q$ can be qualitatively described by assuming that the deficit area is affected by the same acceleration forces throughout the impeller channel. The relation between the accelerations $\hat{a}_m$ and $\hat{a}_q$ is changing on each constant speed line due to the change of the relative stream velocity $\bar{w}$ if the flow is throttled towards lower mass flows.
At operating conditions near stall the relative velocity is low and thus the ratio \( \hat{a}_m/\hat{a}_q \) becomes small. Under these near stall conditions the flow structure is predominated by the Coriolis force and the migration of the secondary flow is controlled by blade-to-blade effects. These effects will force the high-velocity stream to move towards the pressure side while the deficit area migrates towards the suction side. As a result of this movement a flow structure which is similar to the classical Jet-Wake structure is seen. Close to the choke line the highest relative velocities are found and thus the ratio \( \hat{a}_m/\hat{a}_q \) becomes large. Under these conditions the influence of the meridional curvature predominates and as a result the high velocity flow moves towards the hub while the deficit area is drifts towards the shroud.

This qualitative description of the behavior of the impeller flow as a result of the traverse acceleration ratio agrees with the experimental data shown in Figure 4-18 and 4-19. In the experiments a large wake area is found near the shroud wall for high mass flow conditions while near the stall line a weak classical jet-wake structure is seen.

**Application of the theory to the present experimental data**

In Table 4-2 the ratio of the accelerations \( \hat{a}_m/\hat{a}_q \) at impeller exit is computed for different mass flow conditions. The calculation of the acceleration components is performed for the impeller exit (\( R = 200 \text{ mm} \)) and is based upon the area averaged relative velocity \( \overline{w} \) which was measured using the FRAP probe system. The flow is assumed to follow the blade and the effects of slip are neglected. From the geometry of the impeller blade the radii of curvature have been estimated as \( R_{Kq} = 150 \text{ mm} \) and \( R_{Km} = 300 \text{ mm} \), respectively. For this geometrical location the blade angles are \( \varepsilon = 83^\circ \) and \( \beta = 30^\circ \).

For operating conditions near choke acceleration ratios above unity are found while near stall the ratio \( \hat{a}_m/\hat{a}_q \) gets small. The comparison of the different mass flow conditions reveals a strong increase of the acceleration ratio for increased mass flow conditions. This result was expected from the qualitative description of the secondary flow shown above. The analytical description agrees with the experimental finding where at high mass flows an intense migration of the wake area towards the shroud was observed which was not explained by the conventional jet-wake models. It also agrees with the description of other authors (Moore et al. [47 & 48], Inoue et al. [40]) who showed the same dependence of wake location, Rossby number, and flow rate.
The increased acceleration ratio reveals a strong sensitivity of backsweped designs to the influence of the meridional curvature. This leads at high mass flow conditions to a movement of the high velocity flow towards the hub and a drift of the deficit area towards the shroud. For the current case the increase of the wake area near the shroud is even stronger than what can be predicted in the model because the flow is additionally influenced by the tip clearance flow. The tip leakage flow is adding additional deficit flow into the deficit area at the shroud. This mechanism is not accounted for by the model and leads to a further increase of the low work zone.

Table 4-2  Acceleration ratio estimated at impeller exit based upon measurements in the baseline case

<table>
<thead>
<tr>
<th>Mass flow conditions at “STR 17.2” Mu=0.6</th>
<th>Mean relative velocity $\overline{\dot{w}}$</th>
<th>Acceleration ratio $\dot{a}<em>{m}/\dot{a}</em>{q}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>(A) PHI = 0.062 (near choke)</td>
<td>137 m/s</td>
<td>1.33</td>
</tr>
<tr>
<td>(B) PHI = 0.055</td>
<td>126 m/s</td>
<td>1.09</td>
</tr>
<tr>
<td>(C) PHI = 0.051 (design)</td>
<td>120 m/s</td>
<td>0.98</td>
</tr>
<tr>
<td>(D) PHI = 0.042</td>
<td>107 m/s</td>
<td>0.81</td>
</tr>
<tr>
<td>(E) PHI = 0.035</td>
<td>97 m/s</td>
<td>0.71</td>
</tr>
<tr>
<td>(F) PHI = 0.022 (near stall)</td>
<td>82 m/s</td>
<td>0.59</td>
</tr>
</tbody>
</table>
4.7 Variation of the relative tip clearance width

In this section tip influence of the tip gap on the flow structure is studied at the inlet of the vaneless diffuser. In particular, the diffuser configurations “STR 17.2” with the baseline clearance is compared to reduced clearance configuration “STR 15.7”. The geometries of both clearance configurations are described in detail in Section 2.3. The operating conditions which are investigated for the reduced relative clearance case are marked and labeled in Figure 4-25. For comparison also the operational behavior of the baseline case is plotted. The pressure ratio is substantially higher for the smaller tip gap configuration. The comparison of both relative tip clearance cases will be done for similar flow coefficients.

![Figure 4-25](image)

*Figure 4-25* Operating conditions for time resolved FRAP measurements at large clearance configuration “STR 17.2” and small clearance configuration “STR 15.7”
Comparison of the time-averaged velocities for different tip gaps

In Figure 4-26 a comparison of the time-averaged radial and absolute tangential velocities is shown for the large and small tip gap configuration. To enable a comparison all velocities are plotted to the same scale and non-dimensional axial position.

The comparison reveals differences in the distribution of both velocity components. For all conditions the reduced clearance case shows a more uniform velocity distribution than the wide relative clearance case. The radial velocity is elevated for the reduced clearance case over the complete diffuser width. At high mass flow conditions the radial velocity is increased at the
shroud wall and a fuller profile is seen. No flow reversal is encountered. The rise in the radial velocity is partially due to a smaller flow area for the small clearance case but is also an indication of a reduced blockage near the shroud. Generally, the flow profiles for the reduced clearance agree with the flow profiles given in the open literature for industrial compressors (Figure 4-4).

The absolute tangential velocity is higher for the reduced clearance case at all mass flow conditions. Especially towards the shroud higher values of tangential velocity are found as losses in the tip region are reduced. The change in tangential velocity is equivalent to a higher Euler work input into the fluid which results in the higher pressure rise for the small clearance configuration. Towards the stall line only little differences are seen.

These comparisons indicate that the flow pattern is dominated at high mass flow conditions by the tip clearance flow while at near stall conditions other phenomena like inlet tip recirculation and a consequent breakdown of the aerodynamic loading are dominating the flow structure.

In Figures 4-27 and 4-28 the time-resolved radial and absolute tangential velocities are plotted for the reduced clearance case. As in the baseline clearance case discussed above, no clear Jet-Wake structure is observed. In the plots of the absolute tangential velocity a region of high velocities is seen which was identified as the channel wake in the LDA measurements. Compared to the large clearance case this channel wake zone is shifted towards the suction side of the blades. This position of the channel wake corresponds to the location where based on the classical Jet Wake model a wake region is expected. At high mass flow conditions the radial velocity is dominated by the skew of the flow but no local flow reversal is seen as in the large clearance case. Near the stall line a flow profile with little axial variation appears.
Figure 4-27 Radial velocity at R/R2 = 105% for the reduced tip gap configuration “STR 15.7” at different flow coefficients.
4.7 Variation of the relative tip clearance width

Figure 4-28  Absolute tangential velocity at R/R2 = 105% for the reduced tip gap configuration “STR 15.7” at different flow coefficients
Change of the secondary flow pattern at reduced tip clearance

In contrary to the large clearance case discussed above no tip clearance vortex is seen in the velocity contour plots. The absence of the tip clearance vortex is also seen in Figure 4-29 where flow perturbation vectors are plotted. The vector lengths are scaled to the same scale as for the examples shown above for the large clearance case. In general smaller flow perturbations are observed. The secondary flow pattern is dominated by the boundary layer skew along the blades and the channel vortex. Towards low mass flow conditions the flow pattern appears mixed and no channel vortex is seen either.

(A) PHI = 0.062; Near System Resistance Line

(C) PHI = 0.051; Design Incidence

(E) PHI = 0.035

(F) PHI = 0.022; Near Stall Line

Figure 4-29 Secondary flow structure at R/R2 = 105% for the reduced tip gap configuration “STR 15.7” at different flow coefficients
Appendix 4-A  Comparison of LDA and FRAP measurements

At a rotational speed of $\text{MU} = 0.6$ and a specific mass flow coefficient $\text{PHI} = 0.051$ measurements are available at the same location for FRAP and LDA. In Figure 4-30 a comparison of the time averaged radial and absolute tangential velocity is given. One set of data has been measured by the intrusive FRAP probe while the other set was measured by the optical LDA system. A good agreement is seen in the comparison of both measurement techniques. In the FRAP system the velocities are slightly lower which is due to a combination of an underestimation of the unsteady flow temperature in the post processing of the FRAP data as a result of heat transfer through the shaft of the probe and slip of the seeding particles in the optical measurements.

![Figure 4-30 Comparison of LDA and FRAP: Circumferentially averaged absolute tangential and radial velocities (PHI = 0.051 (C))]({})

The contours of absolute tangential and radial velocities obtained by the both measurement systems are plotted in Figures 4-31 and 4-32. The flow features identified in Chapter 4.2 from the LDA measurements are also identified in the FRAP measurements. The channel wake region appears more pronounced in the LDA measurements than in the FRAP measurements. A similar behavior of the blade wake is seen in both sets of measurements. In the FRAP results the tip clearance flow appears stronger than in the LDA measurements and has migrated further towards the hub. The LDA measurement of the radial velocity shows a stronger shear between the tip gap fluid and the channel wake and slightly higher radial velocities are found at the hub wall. In general, the contours obtained with the LDA system are smoother than the contours
obtained with FRAP. This indicates a higher bandwidth and less smoothing during the ensemble averaging process for the FRAP technology. The comparison of the flow structure acquired by both measurement techniques reveals a good agreement. The principal flow structures are observed by both measurement systems and similar velocity contours are obtained. From these comparisons it is concluded that measurements of the flow field using both different measurement set-up can be interchanged. This comparison is confirming the study of Gizzi et al [29] who also found good agreement of FRAP and LDA measurements.

**Figure 4-31** Comparison of the absolute tangential velocity measured with the FRAP and LDA systems at $R/R_2 = 105\%$; $PHI = 0.051$ (C)

**Figure 4-32** Comparison of the radial velocity measured with the FRAP and LDA systems at $R/R_2 = 105\%$; $PHI = 0.051$ (C)
5 Influence of Tip Clearance Width on Stability

In the Chapter 3 the stability of the system was discussed based on the information gained at the inlet of the diffuser. Stability or instability was understood as phenomena which originates somewhere within the compressor system and are affecting the hole compressor system. In Chapter 4 a description of the flow field in the diffuser was shown. It was found that the main impact factor on the performance is the tip clearance width. But is was also found that the flow pattern in the diffuser is dramatically changing with mass flow rate.

In this chapter a detailed description of the phenomena in the impeller which lead into instability is given. Time resolved pressure measurements along the casing are used as a indicator for the behavior of each sub-component of the compressor at different mass flow conditions. The influence of the relative tip clearance on the blade loading and the stability is described. In the measurements a regions of high pressure deviation is seen which is originating at the leading edge of the main blade and is convecting downstream. This feature is interpreted as the trajectory of a tip vortex. The trajectory of these tip vortices and their downstream convection is strongly affected by the mass flow coefficient. If the mass flow is sufficiently small the trajectory of the tip vortex becomes perpendicular to the axis of rotation and the tip vortex is interacting with the adjacent blade. A stability analysis revealed that at this mass flow condition inlet tip recirculation is triggered. If the flow is throttled further, the tip vortex vanishes and rotating stall is initiated in the diffuser. In a comparison of data obtained at different tip clearances it is found that for increased relative tip clearances stronger clearance vortices are induced at the leading edge.

Influence of tip clearance on compressor stability

In centrifugal compressors only little work on the modelling of the tip clearance flow and its influence on stability has been done. Eckardt [22] presented a general description and flow model of the secondary flows within a centrifugal compressor which included effects due to tip clearance flows. Senoo and Ishida [78] described the deterioration of the compressor performance due to tip clearance. Hong et al. [35] showed experimental results on the influence of the relative tip clearance on the flow structure and efficiency in radial compressors. A comprehensive review on tip clearance effects in centrifugal compressors was given by Sitaram and Sridhara [82].
Cumpsty [12] gave a summary of viscous effects in axial compressors and described the flow in the endwall regions as the most important but still least well understood part in compressor flows. He presented a pictorial representation of the three dimensionality and effects in the endwall region (Figure 5-1). In this figure the strengthening of the tip vortex with meridional length and its migration towards mid-channel is seen.

![Figure 5-1 Schematic representation of some viscous effects in axial compressors (from Cumpsty [12]; Page 332)](image)

In the early 1990’s a variety of authors (Chen at al. [8], Storer and Cumpsty [87], and Inoue et al. [41]) presented experimental and numerical studies on the impact of tip clearance flows on the stalling behavior of axial compressors and turbines. Based upon pressure measurements in linear cascades they presented models on the rolling-up of the tip leakage flows into vortices. Recently, Yoon et al. [97] used a dense array of flush-mounted pressure sensors to measure the static pressure fluctuation and its deviation to detect clearance vortices in axial compressors.

**Modeling of tip clearance flows**

One of the earliest and ground braking studies on the modelling of clearance flows was performed by Rains [63] in 1954. He proposed a model to calculate the tip clearance flow and its kinetic energy based upon incompressible potential flow theory. In the study of Rains a simplified and idealized flow geometry is used (Figure 5-2.A). Since the pioneering study of Rains many numerical and experimental studies have been conducted by many authors and the model was improved (Moore et al. [60] and Storer et al. [87]). Denton [16]
summarized different mechanisms of the flow and mixing in the clearance gap of thick and thin blades (Figure 5-2.B&C). In turbines the blades are thick and the residual time in the tip gap region is sufficiently long to mix the clearance flow while in the case of thin compressor blades an isentropic clearance jet is reported. For each of these cases the effect of blockage due to separation was described by introduction of a discharge coefficient $C_D$.

The discharge coefficient $C_D$ relates the actual flow rate through the gap and the theoretical value based upon the pressure difference across the blade. It is closely related to the contraction coefficient $\sigma$ which describes the reduction of the tip gap width by the blockage due to separation in the gap. A theoretical value of the contraction coefficient $\sigma = 0.61$ perpendicular to the chamber line is calculated by Wadia et al. [94] based upon the 2-dimensional flow theory of Rayleigh. For the actual discharge coefficient Moore and Tilton [60] gave a value of $C_D = 0.84$ while Storer and Cumpstey [87] found the best empirical fit to their measurements at a discharge coefficient of $C_D = 0.80$.

For calculating the losses due to tip clearance flow Denton [16] suggest to apply a similar mixing-loss model as for the mixing of a jet in free stream. In this mixing model the kinetic energy transported by the clearance flow is lost in the mixing process with the channel flow. Subsequently, losses caused by the clearance flow can be estimated and a useful measure for the efficiency decrease can be given. It is found, that the primary influence onto the leakage flow is the geometry and the pressure difference across the blade.

The methods described above are not able to provide insight into the details of the clearance flows or the tip vortex nor to calculate its trajectory. More complex models are needed to describe details of the vortex interaction with
the channel flow and the resulting flow structure. Lakshminarayana [56] reviewed and improved the analytical models. He accounted for the presence of the vortex core and introduced a flow model to compute the secondary flow velocity field stemming from these vortex systems. For computation of the loss in efficiency the analysis requires empirical data on the vortex circulation and size. A more universal model of the tip vortex is presented by Chen et al. [8] who provide a scaling approach to calculate the parametric dependence of the vortex trajectory and its strength. A complete prediction of the flow field is only possible by numerical methods which solve the 3-D Reynolds-averaged Navier-Stokes equations (Shum et al [79], Eum et al.[24]). For an accurate estimate of the clearance flows and their influence on compressor stability unsteady computations need to be performed on a very fine grid and thus quite a big amount of computational resources are needed.

5.1 Experimental set-up and data evaluation

An experimental investigation on the pressure fluctuation and the influence of the relative tip clearance on the onset of instability was performed at two relative tip clearances for a rotational speed of \( \mu = 0.6 \) (9949 RPM). In Figure 5-3 the compressor operating lines are shown in terms of the specific work and flow coefficients for the tip clearance ratios \( CR = 4.5\% \) and 12.7%.

![Figure 5-3 Investigated operating conditions](image-url)
The letters in Figure 5-3 mark the operating conditions where detailed pressure measurements have been obtained by a combination of static wall pressure taps and time resolved high frequency response pressure plugs along the casing. The onset of instability is defined as described in Section 3.2 by investigating the frequency content of the pressure fluctuation in the inlet of the diffuser and is marked for each tip clearance case by a black line. With increasing tip clearance a reduction of pressure rise and a shift of the stall line towards higher mass flows is observed. This shift of the stall limit at different clearances agrees with previous findings in axial and radial compressors.

A total of 30 high frequency response sensors are located in the inlet pipe, along the shroud wall and the diffuser. For each measurement location a time series of 300,000 pressure readings has been obtained at a sampling rate of 100 kHz. A more detailed description of these high frequency response sensors is given in Section 2.8. The steady reference pressure for each sensor and the steady pressure along the casing is acquired with a multi channel pressure scanner system.

Using this experimental setup the time history of the static wall pressure can be reconstructed as the sum of the steady pressure $\tilde{p}$, the periodic pressure fluctuation $p'$ and the non-deterministic pressure $p_t$:

$$p_{(t)} = \tilde{p} + \tilde{p}_{(t)} + p'_{(t)}$$

(5.1)

The steady pressure $\tilde{p}$ is acquired by the steady pressure measurement system. The time related pressure fluctuation $\tilde{p}_{(t)} + p'_{(t)}$ is given by the time resolved measurements. By performing an ensemble average on the time resolved pressure reading the periodic fluctuation $\tilde{p}_{(t)}$ due to the blade passing can be separated from the non-deterministic part $p'_{(t)}$. The ensemble averaging is performed based on the impeller position delivered by a pickup on the shaft and all 7 full-blade channels are treated similarly. By calculating for each sensor the phase locked periodic pressure fluctuation $\tilde{p}_{(box)}$ relative to the position of the blades the blade loading distribution throughout the flow path is obtained. One full-blade passage is discretized into 103 equal parts representing an angular discretisation of 0.5 degree. In each of these discrete angular positions or so-called “boxes” more than 2500 samples are acquired and averaged to obtain a good statistical accuracy.

---
9. PSI 9016, Pressure Range 15 PSI true differential; ±0.05 % Full-Scale Accuracy
To allow for a better comparability of different measurements the pressure values are normalized with the inlet static pressure $p_1$ and the normalized phase locked pressure coefficient is obtained:

$$\tilde{C}_{p(box)} = \frac{\tilde{p}_{(box)}}{p_1} \quad (5.2)$$

The non-deterministic contribution to of the pressure fluctuation $p'_{(i)}$ is described by the Root-Mean-Square (RMS) or standard deviation of the pressure samples in each individual angular box. The phase locked non-deterministic level of pressure deviation is computed as:

$$\sigma_{(box)} = \sqrt{\frac{1}{N} \sum_{i=1}^{N} (p_{i(box)} - \tilde{p} - \tilde{p}_{(box)})^2} \quad (5.3)$$

where $N$ is the number of samples and $p_{i(box)}$ are the individual pressure values. The local value of the pressure deviation is interpreted as the level of unsteadiness of the flow due to vortices at this location. It can be used to resolve the trajectory of tip vortices or the extend of separated zones due to incidence.

### 5.2 Steady pressure distribution

In Figures 5-4 and 5-5 the steady wall pressure distribution along the meridional length is plotted for the design tip clearance case “STRAIGHT 17.2” at a clearance ratio of CR = 12.7%. In Figure 5-4 the measured pressure is normalized using the inlet static pressure while in Figure 5-5 the downstream static pressure is used as a reference. Each individual line in the plots represents a different mass flow condition between the choke and the stall limit. Operating conditions in stall are displayed in a different color. For high mass flow conditions a decompression takes place due to acceleration in the inducer section of the impeller. Pressure is built up in the radial section and in the diffuser. Based on quasi three dimensional flow field calculations Gyarmathy et al [33] explain the decompression at the leading edge with a local separation. Towards stall no decompression is observed and the pressure is monotonously rising throughout the machine.
The decompression for high mass flows is also seen in the second plot where the steady pressure distribution is normalized by the exit static pressure (Figure 5-5). Furthermore a similar pressure rise in the radial section of the impeller and the diffuser is observed for all mass flow conditions. The slope of the pressure rise in the diffuser is seen to be nearly independent of the flow rate. The overall pressure rise in the machine is mostly dependent on the pressure rise within the impeller. For unstable operating conditions a slight deterioration of the diffuser performance is seen. This deterioration occurs due to increased blockage in the diffuser by the rotating stall pattern.

For stalled operating conditions a pressure rise in front of the leading edge is encountered. This pressure rise is a sign that energy has been imparted into the fluid. Since this is seen before the impeller leading edge it indicates the occurrence of a local flow recirculation which is transporting energy upstream of the impeller. Gyarmathy et al [33] concluded from similar measurements that a ring-shaped separation bubble exits at impeller leading edge in which fluid is driven around the annulus at blade speed. In this separated zone energized fluid is able to migrate upstream and is seen as raised pressure in front of the impeller. The separation bubble was further investigated by Ribi [64] who found the inlet tip recirculation pattern to occur at the same specific flow coefficient for different impeller speeds.

![Figure 5-4 Steady Wall Pressure Distribution for design clearance ratio (CR = 12.7%); Normalized with the inlet pressure](image_url)
5.3 Blade passing related periodic pressure fluctuation

The normalized periodic pressure fluctuation \( \tilde{C}_{p(box)} \) caused by the blades and its deviation \( \tilde{\sigma}_{(box)} \) are calculated for each sensor along the flow path and are displayed in the representation shown in Figure 5-6. Displayed is a development of the casing surface for one blade pitch. The arc length and the meridional length are plotted on the ordinate and abscissa, respectively. The inlet, impeller, and diffuser sections are labeled. The positions of the full and splitter blade tips are marked, and the dotted lines give the positions of the individual pressure sensors. The obtained values of the normalized phase locked pressure coefficient \( \tilde{C}_{p(box)} \) and the phase locked deviation \( \tilde{\sigma}_{(box)} \) are interpolated along the blade line and are plotted as a color surface on top of this representation.

In Figure 5-7 the phase locked pressure coefficient \( \tilde{C}_p \) is shown at four mass flow conditions for the baseline tip clearance case “STR 17.2”. The color scale is identical for all operating conditions and the specific flow coefficient is labeled in each sub-plot. Positive values of the phase locked pressure coefficient are seen on the pressure side, while negative values are found on the suction side.
The difference across the blade can be interpreted as the absolute value of the blade loading. The integral value of this absolute blade loading is related to the overall power consumption of the compressor. As the mass flow rate is much higher for the case near choke (A) also the power consumption is higher even though the pressure rise is lower. Therefore, despite the lower pressure ratio the highest variations of the pressure coefficient $\tilde{C}_p$ are seen near choke conditions while the smallest absolute blade loading is seen near stall condition (G).

In the high mass-flow cases (A&B) the pressure difference across the blades is maintained in the diffuser. This pressure traces can be interpreted as the trajectory of the Jet-Wake flow leaving the impeller. If the mass flow rate is throttled the trajectory becomes flatter and weaker.
Figure 5-7 Periodic pressure coefficient for the baseline clearance case “STR 17.2” (CR = 12.7%) at different mass flow rates
5.3 Blade passing related periodic pressure fluctuation

For near stall operation condition (G) the flow in the diffuser is disturbed and no clear pattern can be identified in the diffuser. This agrees with the findings in Section 4.5. In these measurements the Jet-Wake pattern was only found for design incidence condition while at low flow rates a stirred up flow profile was found. At all mass flow rates the highest variation of the phase locked pressure coefficient $\tilde{C}_p$ is seen at about 70% to 80% of the meridional length in the radial section of the impeller. For the condition near stall (G) the axial inducer section is barely loaded while in the design incidence condition (B) a more even distribution of the blade loading along the meridional blade length is obtained. The unloading in the inducer section is consistent with the inlet tip recirculation postulated for low mass flows conditions. In the condition (E) a locally high loading is seen at the main blade leading edge. This locally high loading cased by positive incidence.

**Blade loading distribution**

A more quantitative description of the loading along the meridional length of the blade can be given if the pressure difference across the blade is calculated. The blade loading is determined as the difference of the periodic pressure fluctuation value at the pressure and the suction side. The local pressure values are interpolated from the pressure fluctuation field at a circumferential position, which is shifted by three degrees from the blade chamber line. In Figure 5-8.a these positions are marked and labeled by thin lines beside the blades. The resultant blade loading is plotted in Figure 5-8.b. The blade loading is highest at 70 to 80% of the meridional length which agrees with the location of the highest loading calculated in the design phase (Figure 2-10).

![Locations for the calculation of the blade loading](image)

(a) Periodic Pressure Coefficient  (b) Blade Loading Distribution

*Figure 5-8  Designation of the blade loading (CR = 12.7%, PHI = 0.051)
In Figure 5-9 the normalized blade loading distribution is plotted for the baseline clearance case (CR = 12.7%). The small perturbations of the curves are due to the limited meridional resolution of the pressure measurements as only 30 sensors have been placed along the flow path. As explained above due to the higher mass flow rate the absolute blade loading is elevated at near choke condition (A). In the design incidence case (B) a steady distribution of the blade loading is seen. In the case near choke a negative loading is seen as a result of the negative incidence at the leading edge of the full blade. The opposite trend is seen in case (E) where due to positive incidence a locally high loading is seen at the leading edge. In the case near stall only little loading is seen due to the recirculation pattern and no incidence effects can be identified in the inducer section.

(A) PHI= 0.062; Near Choke

(B) PHI= 0.051; Design Incidence

(E) PHI= 0.035

(G) PHI= 0.024; Near Stall

*Figure 5-9* Normalized absolute blade loading for the baseline clearance case “STR 17.2” (CR = 12.7%) at different mass flow rates
5.4 Non-deterministic deviation of the pressure fluctuation

More insight in the stability behavior and the sequence of the events is gained if the deviation of the phase locked pressure fluctuation is investigated. The deviation is calculated from the periodic pressure fluctuation using Equation 5.3. High values of the non-deterministic pressure deviation indicate regions where the temporal fluctuations for the phase locked pressure value are large. Thus, this variable is used as an indicator for areas of high unsteadiness and flow interaction. This is occurring e.g. if a vortex is shed at a different frequency as the blade passing frequency which was used in the phase locked averaging or if a stall cell is present in the diffuser.

In Figure 5-10 the plots of the non-deterministic pressure deviation are given for eight mass flow conditions at a clearance ratio of CR = 12.7% (“STR 17.2”). Sub-plot (A) is acquired at a mass flow rate close to the system resistance line, while sub-plot (H) represents a condition within rotating stall. As a dominant feature a strong pressure deviation is found at the location of the blades. The highest values of the phase locked pressure deviation are found for the highest mass flow coefficient, as in this condition the absolute pressure difference between the pressure and the suction side is largest (Figure 5-7). For smaller flow coefficients the non-deterministic pressure deviation across the blades in the inducer section is diminishing. The clearance flows are seen as high levels of flow unsteadiness beside the location of the blades. Towards low flow coefficient the pattern is dominated by the unsteady phenomena like rotating stall and inlet tip recirculation.

In sub-plot (A) - (E) of Figure 5-10 a pattern of high levels of the non-deterministic pressure deviation is seen at the leading edge of the main blade. In the plots the pattern is marked by a dashed line. This pattern of high non-deterministic pressure deviation is interpreted as the trajectory of the tip vortex. The tip vortex starts in all cases at the leading edge tip of the full-blade and convects with the flow. For smaller mass-flows, it is observed that the tip vortex trajectory moves away from the suction side and migrates towards the adjacent pressure side. A similar behavior of the tip vortex is observed in axial compressors. Yoon et al. [97] and Inoue et al. [41] report on experimental results in axial compressor where they also found a strengthening and upwards movement of the tip vortex at smaller flow coefficients. This was also postulated in a theoretical prediction by Chen et al [8] who linked for axial compressors the upwards migration at decreasing mass flow conditions with a strengthened vortex core and longer convection times.
If the flow is throttled further, the tip vortex hits the leading edge of the adjacent blade (Figure 5-10.F&G) and a strong increase of the non-deterministic pressure deviation in the inlet region of the compressor is seen. This sudden increase is occurring at the mass flow condition were based upon the stability analysis and the steady pressure distribution shown above an inlet tip recirculation zone was predicted. From this coincidence it is concluded that inlet tip recirculation is triggered as soon as the tip vortex is approaching the tip of the adjacent blade. Due to the inlet tip recirculation the overall level of pressure unsteadiness increases in the whole impeller passage. If the compressor is throttled further to a flow coefficient of PHI = 0.02 rotating stall is initiated in the vaneless diffuser (Figure 5-10.H). As a result of the rotating pattern high pressure deviations are recorded in the diffuser section. During rotating stall the inlet tip recirculation zone recovers and low levels of pressure deviations are seen in the inlet. Form this is it concluded that rotating stall is the dominant feature and is able to suppress the recirculation in front the impeller.

Near the suction side of the splitter blade a strong feature can be identified in the sub-plots (A) - (D) of Figure 5-10. The feature is strongest and closest to the suction side for the highest flow coefficient and is weakening with smaller mass flow rates. For smaller mass flow rates it is lifting off the blade surface while at choked condition it appears directly at the suction side. The feature could not be observed for operating conditions in the unstable regime (Sub-plots (F)-(H)). This very strong structure is associated with the tip vortex forming at the leading edge of the splitter blade. In the measurements a strong, but only local, interaction is seen at the splitter leading edge while no downstream convecting trajectory of the splitter blade tip vortex is seen. This is a result of the different interaction mechanism of the splitter blade tip vortex. In the case of the main blade the tip vortex is interacting with the non-rotational free stream approaching the compressor. Behind the splitter blade the vortex is interacting with a skewed and rotating channel flow. As no downstream convection of the tip vortex is seen it is postulated that due to the interaction with the skewed channel flow the tip vortex is entrained in the channel and is lifting off the casing. This assumption is supported by the time resolved flow measurements presented in Chapter 4 where the area which is affected by the tip clearance flows appears shifted towards mid height of the diffuser channel.
5.4 Non-deterministic deviation of the pressure fluctuation

Figure 5-10  Non-deterministic deviation of the periodic pressure fluctuation for the baseline clearance case “STR 17.2” (CR = 12.7%)
Figure 5-10 (Continued) Non-deterministic deviation of the periodic pressure fluctuation for the baseline clearance case “STR 17.2” (CR = 12.7%)
5.5 Comparison of the pressure fluctuations for baseline and reduced clearance

In the plot for the operating conditions (Figure 5-3) it was seen that for smaller clearances an improved operating range is achieved. Within this section the phase-locked pressure coefficient and the pressure perturbation is compared for the baseline and the reduced clearance case.

In Figure 5-11 the normalized steady wall pressure along the meridional length is plotted for the baseline tip clearance case “STR 17.2” and the reduced clearance case “STR 15.7”. The measured pressures are normalized with the inlet static pressure. For each configuration three mass flow rates between choke and the stall limit have been plotted. The measurements in the baseline configuration are displayed using a solid line, while the reduced clearance case is plotted using a dashed line. For all mass flow rates no differences in the steady wall pressure distributions are seen for the inlet and the inducer section. In the radial part of the impeller a significantly better performance is seen for the reduced clearance case. The improved impeller pressure ratio persists throughout the diffuser and is seen as an improved compressor performance in the operating map (Figure 3-19).

![Figure 5-11](image-url)  
*Figure 5-11* Measured steady wall pressure distribution for baseline clearance ratio (CR = 12.7%) and the reduced clearance ratio (CR = 4.5%); Normalized by the inlet static pressure
Comparison of the phase-locked pressure coefficient and blade loading

In Figure 5-12 the normalized blade loading distribution along the main and splitter blade is plotted for four operating conditions at the reduced clearance case “STR 15.7” (CR = 4.5%) while in Figure 5-13 the normalized pressure difference \( (p_{PS} - p_{SS})(z)/p_1 \) across the main blade and the splitter blade is shown. These variables are compared to Figures 5-7 and 5-9 where the same variables are plotted for the baseline case “STR 17.2” (CR = 12.7%) at the same flow coefficients.

At similar mass flow rates the absolute blade loading is higher for the reduced clearance case than for the large clearance case. This is a result of the increased pressure rise seen in the comparison of the operating characteristics Figure 5-3). As in the baseline clearance case a slightly smaller loading is seen on the splitter blade for small mass flow rates while towards choke a high loading of the splitter leading edge is found. The same incidence effect is seen for both clearance cases: Due to positive incidence in the mid range condition (E) a high loading is seen at the main blade leading edge while at the condition near choke (A) the negative incidence is causing a negative loading. Also, for the condition near stall (B) a very low loading is seen for both clearance cases in the inducer section while the radial section is still loaded.

This comparison does not show a significant difference of the loading distribution and does not explain the large differences in the pressure rise between the two clearance cases.
5.5 Comparison of the pressure fluctuations for baseline and reduced clearance

Figure 5-12  Periodic pressure coefficient for the small clearance case “STR 15.7” (CR = 4.5%) at different mass flow rates
Influence of Tip Clearance Width on Stability

Comparison of the non-deterministic pressure deviation at two clearances

More insight into the stability mechanism is gained if the non-deterministic pressure deviation is compared for both clearance cases. In Figure 5-14 the deviation is plotted for selected operating conditions. A comparison with Figure 5-10 reveals generally higher levels of pressure deviations in the baseline clearance case throughout all mass flow cases. For the design incidence condition (B) the non-deterministic deviation is strongly elevated for the region near impeller exit. In this area the tip clearance has been changed most by the shim and the gap difference is largest. In the inlet region a similar trajectory of the tip vortex core is found for both clearance cases.

Figure 5-13 Normalized absolute blade loading for the reduced clearance case “STR 15.7” at different mass flow rates

Comparison of the non-deterministic pressure deviation at two clearances

More insight into the stability mechanism is gained if the non-deterministic pressure deviation is compared for both clearance cases. In Figure 5-14 the deviation is plotted for selected operating conditions. A comparison with Figure 5-10 reveals generally higher levels of pressure deviations in the baseline clearance case throughout all mass flow cases. For the design incidence condition (B) the non-deterministic deviation is strongly elevated for the region near impeller exit. In this area the tip clearance has been changed most by the shim and the gap difference is largest. In the inlet region a similar trajectory of the tip vortex core is found for both clearance cases.
5.5 Comparison of the pressure fluctuations for baseline and reduced clearance

Figure 5-14 Non-deterministic deviation of the periodic pressure fluctuation for the small clearance case “STR 15.7” (CR = 4.7%)
Figure 5-14 (Continued) Non-deterministic deviation of the periodic pressure fluctuation for the small clearance case “STR 15.7” (CR = 4.7%)
For throttling the mass flow (Condition (B&F)) the tip vortex trajectory moves upstream and towards the adjacent pressure side for both clearance cases and finally inlet tip recirculation is provoked (Condition (F)). Differences are seen in operating condition (H) where rotating stall in the diffuser is present for the large clearance case but for small clearance only little deviations are found in the diffuser. This complies with the findings of the general stage behavior in Figure 5-3 where a delay in the stall initiation was found for the small clearance case.

5.6 Modelling of the tip clearance flow

In the previous sections experimental evidence of the influence of the clearance flows on the flow and the stability limit was given. In this section the clearance flows are quantified and the losses associated with the clearance flow will be estimated for different mass flow conditions and clearance widths. In the literature a variety of approaches for describing clearance flows are given. The experimental data have mostly been taken from linear cascades where the clearance height could be altered and measurements of the pressure field inside the airfoil was possible. However, only little work is based on experimental investigations in centrifugal compressors. In the current study a rough estimate of the tip clearance losses shall be given based upon the experimental results presented above. To do so the basic model of Rains and the discharge coefficient derived by Storer et al. [87] and Moore et al. [60] will be applied to the experimental results. As a result a correlation of the leakage flow, the tip clearance width and the operating condition is obtained.

The assumed flow geometry of the tip gap is given in Figures 5-2.A and 5-15. As in the model of Rains an incompressible flow normal to the blade is assumed. Additionally it is assumed that the cordwise velocity of the leakage flow remains unchanged as it passes over the blade and equals the surface velocity on the pressure side \( v_{PS} \) and immediate mixing of the leakage flow with the surrounding flow takes place at the suction side.
By applying Bernoulli’s equation the flow velocity $v_{TC}$ in the tip gap region is calculated based upon the pressure difference over the gap as:

$$v_{TC} = C_D \cdot \sqrt{\frac{2}{\rho} (p_{PS} - p_{SS})} \quad (5.4)$$

In this equation the discharge coefficient $C_D$ is used to deal with the contraction of the flow area due to separation (“vena contracta”). The local pressure difference $(P_{PS} - P_{SS})_z$ is derived from the measurements of the time-resolved wall static pressure fluctuation. In a next step this equation is used to calculate the meridional distribution of the mass flow rate through the gap $dm_{TC(z)}$ as a function of the local clearance width $t(z)$:

$$dm_{TC(z)} = t(z) \cdot C_D \cdot \sqrt{\rho \cdot (p_{PS} - p_{SS})_z} \, dz \quad (5.5)$$

The clearance width is taken from the design gap width shown in Figure 2-4. Once the local mass flow rate is known the overall leakage mass flow and the ratio of the leakage flow and the stage mass flow can be calculated. The leakage ratio sets the overall leakage mass flow rate in relation to the stage mass flow rate:

$$LR = \frac{m_{TC}}{m_{Stage}} \quad (5.6)$$

Based upon the local leakage mass flow Denton [16] suggest to calculate the irreversible mixing loss of the leakage fluid jet with the free stream on the suction side as:
5.6 Modelling of the tip clearance flow

\[
T \Delta s = w_{SS}^2 \cdot \left(1 - \frac{w_{PS}}{w_{SS}}\right) \cdot \frac{dm_{TC}}{m_{Stage}} \tag{5.7}
\]

where \( m \) is the mass flow rate of the main stream while \( w_{PS} \) and \( w_{SS} \) are the relative velocities on the pressure and suction side. This loss function is similar to the mixing loss function applied for cooling jets in turbines and accounts for the loss due to the different velocities of the two fluid bodies. Based upon these considerations the entropy loss in the system due to leakage is coupled to the ratio of leakage flow and the stage mass flow \( dm_{TC}/m_{Stage} \). For current measurements the relative velocities are unknown and thus an entropy loss can not be calculated directly.

Application of the leakage jet model on the measurements

The local mass flow through the clearance space \( dm_{TC} \) is calculated according to Equation 5.5 and is plotted in Figure 5-16 for different mass flow conditions for the baseline clearance case “STR 17.2” (CR = 12.7%) and the case with reduced clearance “STR 15.7” (CR = 4.5%). The integral values of the leakage flow across each individual blade is noted in the figure. In all conditions the largest leakage flow is seen for the large clearance case while only little differences are seen between full and splitter blades. For the reduced clearance case CR = 4.5% a constant local leakage with little variation along the meridional length is seen. In the large clearance case CR = 12.7% the highest leakages are found in the radial section where the highest loading and the largest clearance gap is present.
(B) PHI= 0.051; Design Incidence

(E) PHI= 0.035

(G) PHI= 0.024; Near Stall

LARGE CLEARANCE “STR 17.2” SMALL CLEARANCE “STR 15.7

Figure 5-16 Local leakage flow for the baseline clearance (STR 17.2) and small clearance case (STR 15.7) at different mass flow rates
In Figure 5-17 the overall leakage mass flow rate and the leakage ratio are summarized for eleven operating conditions and both clearances investigated. The overall leakage mass flow rate is obtained by integrating the local leakage flows for all full and splitter blades while according to Equation 5.6 the leakage ratio sets the overall leakage in relation to the stage mass flow rate.

For operating conditions near stall the overall leakage flow is lower than close to choke but as the stage mass flow is reduced the leakage ratio is highest for conditions near stall. A nearly linear relation is found between the leakage ratio and the mass flow. The highest leakage ratio of around 23% stage mass flow is found for the large clearance near stall. This value appears large as fluid which is crossing more than one blade tip in counted more than once.

**Table 5-1 Relation of the clearance ratio CR and the leakage ratio LR**

<table>
<thead>
<tr>
<th>Clearance ratio CR</th>
<th>Leakage Ratio LR near Choke</th>
<th>Leakage Ratio LR near Stall</th>
</tr>
</thead>
<tbody>
<tr>
<td>Large clearance “STR 17.2”</td>
<td>12.7%</td>
<td>13.6%</td>
</tr>
<tr>
<td>Small clearance “STR 15.7”</td>
<td>4.5%</td>
<td>6.4%</td>
</tr>
<tr>
<td>Relation between large and small clearance case “STR 17.2:STR 15.7”</td>
<td>2.8</td>
<td>2.12</td>
</tr>
</tbody>
</table>
In Table 5-1 the relation between the leakage ratios of both clearances is calculated and is compared to the relation of the clearance ratio (Equation 2.6). In this centrifugal compressor a similar relation between the leakage ratios for large and small tip gap is found for operating points close to stall and close to choke, respectively. This relation is slightly lower than the relation between the clearance ratios. This discrepancy is caused by the axial widening of the gap (Figure 2-4) which results in a larger increase of the clearance at the exit than in the inlet region.

5.7 Summary of the stability mechanism

No significant differences have been found in the blade loading distribution for large and reduced relative tip clearance width. But, the level of the non-deterministic pressure deviations reveals a higher flow disturbances for the large clearance case. The increased levels of pressure disturbance are caused by the increased intensity of the tip clearance flows and indicate stronger secondary flows. The onset of rotating stall is shifted to higher mass flows if the clearance is increased while the onset of tip recirculation is not affected by a change in tip clearance width. For increased clearance width stronger clearance flows establish and interact with the main flow. The leakage ratio scales in both cases with the clearance width. It is concluded that the clearance width is the dominant effect on the leakage mass flow and is thus responsible for the losses in the impeller. These increased losses lead to a reduction in pressure rise and stall margin as was seen in Section 3.5 in the comparison of the operational characteristic for both clearance width.

For highest mass flow conditions the flow pattern is dominated by an evenly distributed blade loading from the inlet to the exit across all blades. Highest values are found as predicted by the design calculations near the exit of the impeller. A rolling up tip vortex sheet is found on the suction side of the main blade. Starting on the leading edge, the tip vortex is convected with the flow and is migrating towards the subsequent pressure side. If the flow coefficient is reduced, the migration of the tip vortex increases and the tip vortex trajectory moves upstream. As soon as the tip vortex approaches the subsequent blade inlet tip recirculation is triggered which leads to a sudden increase of random pressure fluctuation in the inducer section and the suction pipe. If the flow is throttled further rotating stall is initiated in the diffuser while the inlet recovers.
6 Discussion and Conclusion

This experimental work investigated the aerodynamic behavior and stability of small-scale highly loaded centrifugal compressors. These compressors, which are used in small-scale distributed power applications and automotive turbocharging, demand the development of strategies for further enhancement of the stable operating margin and performance. A special focus was put on the effects caused by low Reynolds numbers, high blade-loading, and large relative tip clearance as these are common features of compressors in small-scale applications. In the course of this thesis the flow structures and their interactions have been investigated experimentally using an enlarged model of such compressors.

Design and scaling of the compressor stage

A model of a highly loaded compressor was developed in-house and was successfully applied at different scales. The design volume flow rate for the true-scale model of \( \dot{V} = 0.095 \text{ m}^3/\text{s} \) and the design pressure ratio of \( \pi = 2.8 \) was fulfilled at an impeller speed of 133,000 RPM. To enable accurate time and spatially resolved measurements an enlarged research facility was equipped with a geometrically similar compressor stage. The use of the enlarged research facility provided the opportunity to measure very accurately while the true-scale experiment provides the baseline for the comparison of the system behavior. A comparison of the real-scale turbocharger facility and the enlarged research compressor revealed a good agreement. In stable operating conditions matching compressor performance and similar onset of instability was encountered in both scales.

A parametric study of the influence of the Reynolds number on the compressor revealed that, for this compressor, the Reynolds number has only little effect on the compressor performance. A reduction in the pressure rise of 0.5% was found for a reduction in the Reynolds number by a factor of 3.1 while the onset of instability was not changed. This result suggests that, at least for the tested configuration, the dominant instabilities are driven by inviscid processes and do not depend on the scale or the Reynolds number. A parametric study of the effects of relative tip clearance was performed by shimming the compressor casing and thus increasing the tip clearance axially. It revealed a dominant effect of tip clearance width on the compressor performance. For reduced clearance the pressure ratio is improved significantly. The choke and the system resistance limit remain unchanged. For high impeller speeds the
instability limit is not affected while for low impeller speeds an improvement is found for the reduced tip clearance case. The diffuser width had no significant effect on the onset of instability or performance of the compressor stage. It is concluded that in the current case, the reduction in pressure rise reported for the increased tip gap are due to flow effects and the losses in the impeller and do not depend on the diffuser width.

Flow structure in the vaneless diffuser

In order to investigate tip clearance effects, detailed velocity measurements have been performed at different diffuser and tip clearance configurations. Using this data the formation of the diffuser flow structure and its dependence on the flow through the tip gap was studied. The tip clearance width has a dominating effect on the formation of secondary flows in the diffuser. The time-resolved measurements revealed a significant non-uniformity of the flow at the discharge of the impeller. High radial velocities are present near the hub, while near the shroud, even at stable operating conditions, reversed flow was encountered. This results in a severe skew of the flow in the diffuser. The flow structure in this highly loaded compressor does not comply with the classical Jet-Wake pattern. For increased clearance an additional vortical flow feature is identified at the shroud, which destabilizes the compressor and deteriorates the performance. It is proposed that in compressors with large relative tip clearance a modified flow model is more appropriate. This flow model includes tip leakage flows and describes the interaction of the highly vortical clearance flow with the passage flows. The mixing process in the vaneless diffuser takes place on two different time scales. The jet and the blade wake are interacting rapidly at the impeller exit. The clearance flow is replacing the channel wake but both flow zones remain stable throughout the diffuser.

Furthermore, the flow pattern is influenced to a great extent by the changes in the mass flow condition and tip gap. For high mass flow rates the flow in the diffuser is severely skewed while for near stall conditions uniform block profiles are found. A model is used to explain the displacement of the wake as a result of the changed forces on the flow at different mass flow conditions and good agreement with the experiment is found. This altering flow structure is of special interest for the design of vaned diffusers as the user of compressor wishes to operate at a wide range of the mass flow rates. Goals for further developments of compressors are to improve the compressor performance at these off-design conditions. Therefore the knowledge of the flow field at these conditions is of special interest.
Off-design operating conditions and stability

The stability mechanism and the effect of clearance width on the onset of instability has been analyzed based upon measurements of the fluctuation of the static wall pressure along the casing. The clearance width has a dominating effect on stability and performance of the compressor. The onset of rotating stall is shifted to higher mass flows if the clearance is increased while the onset of tip recirculation is not affected by a change in tip clearance width. No significant differences have been found in the blade loading distribution for both relative tip clearance widths. But the steady pressure rise seen in the radial section was significantly deteriorated for the enlarged clearance configuration. For the wide clearance case the level of pressure deviations reveal higher flow disturbances. The increased levels of pressure disturbance are caused by the increased intensity of the tip clearance flows and indicate stronger secondary flows and higher mixing losses. These increased losses lead to the observed reduction in pressure rise and stall margin for the wider clearance configuration. In both clearance cases the trajectory of a leakage vortex could be identified in the deviation of the phase locked pressure fluctuation at the main blade suction side. This leakage vortex is a result of the interaction of the leakage jet and the undisturbed passage flow. On the splitter blade an immediate interaction between the clearance and the skewed channel flow is seen but no trajectory of a leakage vortex could be identified.

Summary of the stability mechanism

For high mass flow rates the blades are evenly loaded from the inlet to the exit across all blades. Highest blade loading is found as predicted by the design calculations near the exit of the impeller. A rolling up tip vortex sheet is found on the suction side of the main blade. Starting on the leading edge, the tip vortex is convected with the flow and is migrating towards the adjacent pressure side. If the flow coefficient is reduced, the apparent stagger angle of the leakage vortex is increased and the tip vortex trajectory moves upstream. If the mass flow rate is sufficiently small, the tip vortex interacts with the adjacent blade row. A stability analysis revealed that at this conditions inlet tip recirculation is triggered. This leads to a sudden increase of random pressure fluctuation in the inducer section and the suction pipe. If the flow is throttled further, rotating stall is initiated in the diffuser while the inlet recovers.
Outlook and suggestion for further work

In the current study the effects of upstream or downstream equipment on the flow structure were not investigated. In real industrial applications the inlet is distorted by bends of the pipework, struts or inlet guide vanes. Downstream of the impeller additional equipment like diffuser vanes or scrolls are present. These distortions might influence the stability of the compressor system and should be investigated in detail. This requires the simulation of these distortions in the research facility and the measurement of the flow field at different circumferential locations.

In the current study no measurement techniques were available to quantify the entropy production or efficiency forfeit due to this interaction. It is suggested to apply novel measurement techniques for the acquisition of time resolved total temperature at different tip clearance configurations.

The investigation of the stability mechanism based upon time-resolved measurements of the static wall pressure fluctuations provided surprising insights into the flow in the impeller passage. It is a easy-to-use technology and allows judgement on the development of stall precursors such as inlet tip recirculation or incidence effects on the splitter blade. But it also provides the opportunity to make a rough estimate on the leakage flow rate and the loss generation in the impeller. It is suggested to make further use of this technology either in parametric studies on clearance effects or tip profiling and for assessment of the stability of the compressor.

Flow measurements in the diffuser revealed an unexpected third flow pattern in addition to the classical Jet-Wake pattern. The detailed flow measurements revealed the existence of a highly vortical clearance flow which is related to the clearance width. This additional clearance flow pattern is interacting with the passage flow and alters the flow structure at impeller exit significantly. Therefore certain design methods might not be valid for the calculation of compressors with large relative tip clearance. It is suggested to review the classical design tools and implement the clearance flow as a third zone in the modelling.
Appendix
A References


Transactions of the ASME, 121(3), 499-509.


B  Frequently used symbols

Symbols

\( a \)  Speed of sound
\( \dot{a} \)  Acceleration vector
\( b \)  Diffuser width
\( f \)  Frequency
\( p \)  Pressure
\( \dot{p} \)  Angular momentum flux
\( t \)  Time
\( t \)  Tip clearance width
\( v \)  Absolute velocity
\( w \)  Relative velocity
\( z \)  Axial Location
\( A \)  Area
\( D \)  Diameter
\( L \)  Length
\( R \)  Radius
\( R_{z_{iso}} \)  Mean surface roughness
\( T \)  Temperature
\( U \)  Tip speed
\( V \)  Voltage

Greek Symbols

\( \alpha \)  Absolute flow angle
\( \beta \)  Relative flow angle
\( \epsilon \)  Meridional flow angle
\( \chi \)  Blade angle
\( \phi \)  Yaw angle
\( \gamma \)  Pitch angle
\( \rho \)  Density
\( \kappa \)  Gas constant
\( \eta \)  Efficiency
\( \sigma \)  Pressure deviation
\( \Omega \)  Rotational Speed
\( \lambda \)  Laser wave length
\( \Theta \)  LDA probe alignment angel
Indices

- $r$: radial direction
- $\theta$: tangential direction
- $z$: axial direction
- $t$: total
- $s$: static
- $e$: Excitation
- $0$: Stage inlet
- $1$: Impeller inlet
- $2$: Impeller Exit
- $5$: Diffuser Exit

Dimensionless Numbers

- $\phi$: Specific flow coefficient
- $\psi$: Specific work coefficient
- $k$: Reduced Frequency
- $\pi$: Total-static pressure ratio
- $Cp$: Pressure Coefficient
- $SM$: Surge-to-Choke stability margin
- $CR$: Tip clearance ratio
- $LR$: Leakage ratio
- $Ha$: deHaller Number
- $Mu$: Impeller tip Mach Number
- $Re$: Reynolds Number
- $Ro$: Rossby Number
- $Tu$: Turbulence Level

Abbreviations

- STR 17.2: Baseline Configuration
- STR 15.7: Reduced Clearance Configuration
- STEP_15.7: Stepped Configuration: wide tip clearance, narrow diffuser
- PINCH 5: Pinched Diffuser with wide tip clearance
- FRAP: Fast Response Aerodynamic Probe
- FPP: Fast Pressure Plug
- LDA: Laser Doppler Anemometry
- PS: Pressure Side
- SS: Suction Side
C Curriculum Vitae

Matthias Wolfgang Schleer
Born on the 9th of May 1973 in Freiburg im Breisgau; Germany

Education

Studies
1994 - 1999: Study of Mechanical Engineering,
Rheinisch-Westfälische Technische Hochschule (RWTH), Aachen

Working Experience
1999 - 2000: Research Engineer, Lehrstuhl für Reaktorsicherheit und -technik,
Rheinisch-Westfälische Technische Hochschule (RWTH), Aachen
2000 - 2005: Research Assistant, Turbomachinery Laboratory,
Swiss Federal Institute of Technology (ETH), Zurich
since Nov. 2005: Development Engineer, ABB Turbo Systems Ltd., Baden