Rotating stall in wide vaneless diffusers

Ljevar, S.

DOI:
10.6100/IR617120

Published: 01/01/2007

Document Version
Publisher's PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:

• A submitted manuscript is the author's version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

Citation for published version (APA):
Rotating Stall in Wide Vaneless Diffusers

PROEFSCHRIFT

ter verkrijging van de graad van doctor aan de Technische Universiteit Eindhoven, op gezag van de Rector Magnificus, prof.dr.ir. C.J. van Duijn, voor een commissie aangewezen door het College voor Promoties in het openbaar te verdedigen op woensdag 14 februari 2007 om 16.00 uur

door

Svetlana Ljevar

geboren te Banja Luka, Joegoslavië
# Contents

## Summary

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 Introduction</td>
<td>1</td>
</tr>
<tr>
<td>1.1 Axial and centrifugal compressors</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Compressor performance</td>
<td>2</td>
</tr>
<tr>
<td>1.3 Rotating stall</td>
<td>4</td>
</tr>
<tr>
<td>1.3.1 Axial compressors</td>
<td>5</td>
</tr>
<tr>
<td>1.3.2 Centrifugal compressors</td>
<td>8</td>
</tr>
<tr>
<td>1.4 Vanless diffuser rotating stall</td>
<td>9</td>
</tr>
<tr>
<td>1.5 Research objectives and approach</td>
<td>11</td>
</tr>
<tr>
<td>2 Numerical model</td>
<td>13</td>
</tr>
<tr>
<td>2.1 Scaling</td>
<td>13</td>
</tr>
<tr>
<td>2.2 Numerical modeling aspects</td>
<td>16</td>
</tr>
<tr>
<td>2.3 Two-dimensional rotating instability</td>
<td>19</td>
</tr>
<tr>
<td>2.4 Stability limit</td>
<td>23</td>
</tr>
<tr>
<td>2.5 Discussion</td>
<td>24</td>
</tr>
<tr>
<td>3 Experimental validation</td>
<td>25</td>
</tr>
<tr>
<td>3.1 Operating flow conditions</td>
<td>26</td>
</tr>
<tr>
<td>3.2 Comparison with the data found in the literature</td>
<td>27</td>
</tr>
<tr>
<td>3.3 Water model</td>
<td>31</td>
</tr>
<tr>
<td>3.3.1 Experimental setup</td>
<td>32</td>
</tr>
<tr>
<td>3.3.2 Measurements</td>
<td>34</td>
</tr>
<tr>
<td>3.3.3 Measurement results</td>
<td>35</td>
</tr>
<tr>
<td>3.3.4 Comparison with the numerical model</td>
<td>36</td>
</tr>
<tr>
<td>3.3.5 Secondary flows</td>
<td>39</td>
</tr>
<tr>
<td>3.4 Vanless diffuser measurements</td>
<td>42</td>
</tr>
<tr>
<td>3.4.1 Experimental setup</td>
<td>42</td>
</tr>
<tr>
<td>3.4.2 Pressure measurements</td>
<td>44</td>
</tr>
<tr>
<td>3.4.3 Flow field measurements</td>
<td>47</td>
</tr>
<tr>
<td>3.5 Discussion</td>
<td>53</td>
</tr>
<tr>
<td>4 Instability analysis</td>
<td>59</td>
</tr>
<tr>
<td>4.1 Two-dimensional inviscid stability analysis</td>
<td>59</td>
</tr>
<tr>
<td>4.2 Impeller startup</td>
<td>60</td>
</tr>
<tr>
<td>Section</td>
<td>Title</td>
</tr>
<tr>
<td>---------</td>
<td>-------</td>
</tr>
<tr>
<td>4.3</td>
<td>Jet-wake interaction</td>
</tr>
<tr>
<td>4.4</td>
<td>Tangential acceleration</td>
</tr>
<tr>
<td>4.5</td>
<td>Shear-layer between in- and outlet flow region</td>
</tr>
<tr>
<td>4.6</td>
<td>Effect of viscosity</td>
</tr>
<tr>
<td>4.7</td>
<td>Discussion</td>
</tr>
<tr>
<td>5</td>
<td>Influence of geometry</td>
</tr>
<tr>
<td>5.1</td>
<td>Diffuser radius ratio</td>
</tr>
<tr>
<td>5.1.1</td>
<td>Comparison with the two-dimensional inviscid flow model</td>
</tr>
<tr>
<td>5.1.2</td>
<td>Comparison with experimental data from literature</td>
</tr>
<tr>
<td>5.1.3</td>
<td>Pressure fluctuations analysis</td>
</tr>
<tr>
<td>5.2</td>
<td>Diffuser width ratio</td>
</tr>
<tr>
<td>5.2.1</td>
<td>Quasi-steady viscous contribution</td>
</tr>
<tr>
<td>5.2.2</td>
<td>Corrected viscous contribution</td>
</tr>
<tr>
<td>5.2.3</td>
<td>Discussion</td>
</tr>
<tr>
<td>5.3</td>
<td>The impeller</td>
</tr>
<tr>
<td>5.4</td>
<td>Discussion</td>
</tr>
<tr>
<td>6</td>
<td>Concluding discussion</td>
</tr>
<tr>
<td>6.1</td>
<td>Conclusions</td>
</tr>
<tr>
<td>6.2</td>
<td>Discussion and recommendations</td>
</tr>
<tr>
<td><strong>Bibliography</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Symbols</strong></td>
<td></td>
</tr>
<tr>
<td><strong>A</strong></td>
<td>Two-dimensional inviscid flow model</td>
</tr>
<tr>
<td>A.1</td>
<td>Flow in the vaneless diffuser</td>
</tr>
<tr>
<td>A.2</td>
<td>Boundary condition at diffuser exit</td>
</tr>
<tr>
<td>A.3</td>
<td>Simplified case</td>
</tr>
<tr>
<td><strong>B</strong></td>
<td>Unsteady fully developed flow model</td>
</tr>
<tr>
<td>B.1</td>
<td>Exact solution of an oscillating channel flow</td>
</tr>
<tr>
<td>B.2</td>
<td>Quasi-steady approach of the viscous contribution</td>
</tr>
<tr>
<td>B.3</td>
<td>Determination of the correction term</td>
</tr>
<tr>
<td><strong>Acknowledgments</strong></td>
<td></td>
</tr>
<tr>
<td><strong>Résumé</strong></td>
<td></td>
</tr>
</tbody>
</table>
Summary

Rotating Stall in Wide Vaneless Diffusers

In the past 50 years a lot of attention has been given to the study of airflow through compressor engines. The main reason for this is that the flow conditions can become unstable when a compressor operates close to its optimal design conditions. At reduced flow rates, stable operating condition of compressors can be disturbed by unsteady flow phenomena such as surge and rotating stall, which may result in a significant drop in aerodynamic performance. Besides limiting the operating range of compressors, these phenomena cause critical operating states with particularly strong dynamical loading on the blades and they usually cause noise nuisance. Therefore, they cannot be tolerated during compressor operation.

This research is focused on the flow dynamics of rotating stall. Several studies indicate that two or more different flow mechanisms might be responsible for rotating stall in vaneless radial diffusers. Generally, one mechanism is associated with the two-dimensional core flow instability occurring in the wide vaneless diffusers when the critical inlet flow angle is reached, and the other mechanism is associated with the three-dimensional wall-boundary-layer instability occurring in narrow diffusers. Since this thesis focuses on wide vaneless diffusers, first the core flow instability is studied without the influence of the wall boundary layers, and later the influence of the wall boundary layers is added to the two-dimensional analysis.

To model the vaneless diffuser flow in the plane parallel to the diffuser walls a commercial software package Fluent is used, in which a standard incompressible laminar flow solver is applied. At the diffuser inlet a rotating jet-wake velocity profile is prescribed, while at the diffuser outlet a constant static pressure is prescribed. The numerical model reveals that in vaneless radial diffusers a two-dimensional rotating instability exists, which occurs when the critical inlet flow angle is reached. The obtained rotating instability has similar characteristics as the rotating stall phenomenon since they both consist of a number of rotating cells, which propagate around the circumference at a fraction of the impeller speed.

For the experimental validation of the two-dimensional core flow model, first a zero-throughflow water model of the wide vaneless diffuser has been built. Using this water model, a similar core-flow instability as obtained by the numerical model is observed at mid-height of the vaneless diffuser space. Furthermore, experiments are also performed in an open-loop radial flow pump stage working with air. This stage approaches the actual environment of the rotating stall phenomenon, but it also meets the assumptions made in the two-dimensional numerical model. The diffuser is wide enough and no volute is used, which means that the diffuser outlet is connected to the space with constant pressure.

Rotating stall phenomena occurring in the wide vaneless diffuser space are analyzed using Particle Image Velocimetry. The experimental rotating stall phenomenon is found to resemble the two-dimensional rotating instability obtained by the numerical model. The size of the rotating cells, their relative distance from the impeller and their propagation speed is nearly the
same as obtained with the numerical model. Only the number of rotating cells is found to be slightly lower, which is probably due to the somewhat higher rate of the viscous dissipation.

The developed numerical model is used to study the characteristics of the two-dimensional rotating instability. To study the physics of the instability, the results are compared with those from a two-dimensional inviscid stability analysis, and the influence of the impeller startup, the jet-wake pattern, the tangential acceleration, the outlet boundary conditions and the turbulent viscosity is investigated. The jet-wake is found to have a negligible influence on the core-flow instability. The tangential forcing is found to play a dominant role in the stability criterion, but due to the short times involved it probably does not cause the two-dimensional rotating instability. Therefore, the shear-layer instability between high velocity fluid near the impeller and the low velocity fluid near the diffuser outlet, is considered to be the primary cause of the two-dimensional rotating instability.

To study the influence of the diffuser geometry on the two-dimensional rotating instability, the number of impeller blades, the diffuser radius ratio and the diffuser width ratio are varied. Also the influence of the impeller and diffuser outlet flow conditions is discussed. The numerically obtained influence of the diffuser radius ratio is compared with the two-dimensional inviscid stability analysis and with the experimental data found in the literature, and quite good agreement is found. In all cases the critical flow angle, the number of cells and their propagation speed are found to have the same trend versus the diffuser radius ratio, and their values are of the same order of magnitude. It is also shown that the numerically obtained static pressure signature during transition from stable to unstable operating flow-condition looks similar to the measured static pressure fluctuations in real vaneless diffusers during rotating stall inception. It is shown that the diffuser geometry as well as the operating flow conditions influence the pressure signals. The influence of the diffuser width is also investigated. To account for the influence of wall-boundary layers, the two-dimensional numerical model is extended to a quasi three-dimensional model. To that end, extra source terms, which represent the estimated shear force generated by the wall-shear layers, are added to the momentum equations. First the steady shear force is applied, and later the influence of the unsteady shear force is investigated. The analysis showed that the resistance to the flow generated by the two wall shear layers will result in a slight decrease of the number of rotating cells.

This study contributes to a better understanding of one of the flow mechanisms that might be responsible for rotating stall in vaneless radial diffusers. It also gives an overview of the vaneless diffuser parameters that influence the core-flow stability and the rotating stall characteristics.
Chapter 1

Introduction

This research originates from the work of Meuleman [50] on measurement and unsteady flow modeling of centrifugal compressor surge and Willems [62] on modeling and bounded feedback stabilization of centrifugal compressor surge. They have designed and implemented a model for active surge control on a small scale centrifugal compressor, and have observed that rotating stall occurs when surge was postponed. In order to develop rotating stall control systems, better understanding of the rotating stall flow dynamics is required. Understanding of the basic effects leading to the circumferential breakdown of the flow is important for successfully delaying or controlling it. Therefore, this project was supported by Siemens Demag Delaval Turbomachinery B.V., who’s interest lies in development of active surge and rotating stall control. This project was also supported by TNO, since they are interested in the aero-acoustic effect of rotating stall, which can cause damage to the systems in which compressors are implemented.

The aim of this chapter is to give an introduction to compressor unsteady flow phenomena and to give an overview of the studies performed by means of a literature survey. First, compressor operating principles and their applications are presented for both axial and centrifugal compressors. Then, compressor performance map and unsteady flow phenomena such as surge and rotating stall are introduced, and a literature survey is given about rotating stall in centrifugal compressors. This chapter ends with an overview of the research objectives and approach.

1.1 Axial and centrifugal compressors

The main purpose of compressors is pressurization of gases for different types of engines and industries. Compressors are used in aircraft engines, industrial gas turbines and turbo-charged combustion engines as well as in process and chemical industries. In these applications power is being produced by expansion of the gases pressurized by the compressor.

Compressors have two main working principles. One working principle is pressurization by decreasing the gas volume, where volume decrease of the trapped gas results in a pressure increase. This working principle is common for the reciprocating compressors in which the compression is realized by a moving piston in a cylinder, but also for the rotary compressors where the compression is realized by rotating lobes and screws or sliding vanes. The other working principle is pressurization by momentum transfer, where gas is first accelerated resulting in a total pressure rise, and then decelerated resulting in a static pressure rise. This working principle is common for rotating compressors such as axial and radial- or centrifugal compressors.

In figure 1.1, the most vital parts of axial and centrifugal compressors are shown. These are
CHAPTER 1. INTRODUCTION

Figure 1.1: On the left-hand side a multi-stage axial compressor and on the right-hand side a centrifugal compressor

the rotor and stator cascade for axial compressors, and the impeller, diffuser and volute for centrifugal compressors. On the left-hand side of figure 1.1, a multi-stage axial compressor is presented consisting of rotor-blade and stator-blade cascades, and on the right-hand side a schematic drawing of a centrifugal compressor is shown. Acceleration of the gas takes place in the rotor for axial compressors and in the impeller for centrifugal compressors, while deceleration takes place in the stator for axial compressors and in the diffuser for centrifugal compressors. The diffuser is always placed after the impeller, as illustrated in the top view of a compressor stage in figure 1.3. The main difference between axial and centrifugal compressors is the process direction. In axial compressors, the operating medium is processed in axial direction, while in centrifugal compressors, the operating medium enters the compressor in axial direction and leaves the impeller in radial direction. The unsteady flow phenomena such as surge and rotating stall occur in both, axial and centrifugal compressors.

1.2 Compressor performance

Each compressor in a gas turbine engine has a compressor map, as illustrated in the diagram in figure 1.2. Compressor maps are important, since they are used for predicting the performance of a gas turbine engine, both at design and off-design conditions. The $x$-axis is usually some function of the compressor entry mass flow. In centrifugal compressors non-dimensional mass flow is given in terms of a flow coefficient $\phi$,

$$\phi = \frac{\dot{m}}{\rho_1 U_2 d_2^2},$$

(1.1)

where $\dot{m}$ is mass flow rate, $\rho_1$ density of gas at the inlet of the compressor, $U_2$ impeller tip speed and $d_2$ impeller outlet diameter. The $y$-axis is usually the pressure ratio ($p_{03}/p_{01}$) where $p_0$ is the stagnation pressure or total head pressure and indexes 1 and 3 refer to the impeller inlet and diffuser outlet, respectively. Quite common for centrifugal compressors is to express the
pressure rise in terms of a normalized static pressure rise $\psi$,  

$$\psi = \frac{\Delta p_{st}}{\rho_1 U_2^2}. \quad (1.2)$$

The strongly curved, near horizontal contours in the main part of the compressor map are constant impeller speed contours. They are a measure for the rotor blade tip Mach number. The impeller speed increases in the direction of the arrow. The slightly curved, near vertical contours in the main part of the compressor map are known as the surge and stall line. These lines indicate the transition area between the stable and unstable operating condition. On the left of these lines, a region of unstable flow is present, which is an area that needs to be avoided.

At low mass flow rates, the compressor performance is characterized by the occurrence of unsteady flow phenomena. Unsteady flow phenomena may appear as high frequency rotating flow separations in the compressor cascades, so called rotating stall, or as low-frequency fluctuations of the mass flow in the whole compressor system, so called surge. These aerodynamic instabilities usually occur at the top of the constant impeller speed contours or near the top where the constant impeller speed contour has a positive slope. In applications that deal with surge, the whole compressor system, including ducting, storage, throttle and compressor itself, has a large influence on the stability of the compressor flow. In contrast to surge, rotating stall is a phenomenon of local character in which oscillations are generated only in one or two components of the compression system. Both forms of instability limit the operating range of compressors, they cause critical operating states with particularly strong dynamical loading on the blades and noise nuisance, and therefore cannot be tolerated during compressor operation. In the establishment of an engine operating line, it is absolutely necessary to know where the compressor surge and/or stall line is.
Rotating stall can be generated by destabilization of the impeller or rotor flow as well as destabilization of the diffuser or stator flow. In case of rotating stall, one or more stalled passages involving a small number of blades rotate around the circumference at a fraction of the impeller or rotor speed. In figure 1.3, the rotating stall phenomenon in a vaned radial diffuser of a centrifugal compressor is illustrated, where three blade passages are stalled. Rotating stall usually propagates in the compressor rotation direction with respect to the absolute frame of reference, and opposite to the rotation direction with respect to the relative frame of reference. One area of stalled passages is called the rotating stall cell, which is characterized by having a reduced or no through flow. Rotating stall is characterized by the number of rotating stall cells, which can be as high as nine or as low as one. Stall usually starts with multiple cells, and as the mass flow rate decreases, the cell number may increase or decrease, but the percentage of the annulus blocked by the total cell area consistently increases. Depending on the compressor configuration, the propagation speed of rotating stall varies from 10 to 90 % of the impeller or rotor speed. Smaller rotating stall cells usually propagate faster than larger ones.

It is possible to operate the compressor during rotating stall because the compressor system can still be stable, but it is risky and not advised to do so. Rotating stall can cause damage and noise nuisance because of large vibratory stresses imposed on the impeller blades. For this reason compressors are currently operated away from their optimal operating conditions increasing both fuel consumption and engine weight. To avoid occurrence of rotating stall, a compressor is operated at reduced pressure ratios to keep a safety margin to the stability limit, which results in loss of high pressure ratios. Rotating stall is one of the most important limitations of the compressor performance, whose effects are well known, but whose origins and
1.3. ROTATING STALL

Most experimental evidence of rotating stall in axial compressors concerns the pre-stall condition, while in centrifugal compressors mostly rotating stall types and interaction between the impeller and diffuser are studied. The studies about rotating stall inception in axial compressors showed that there are three different stall inception patterns.

The first type of rotating stall inception is a short-length-scale disturbance, which is also known as spike. Spike is created by the local stalling of the particular blade row. The local stalling of the blade row including the blockage of the incoming flow is illustrated in figure 1.4, where also the rotation direction of the stalled region is indicated with respect to the relative frame of reference. This disturbance got its name because of the spike-like appearance in the velocity traces. Spike appears suddenly and in its initial phase it is limited to the blade tip region of only one or a few blade passages. Then, it develops within a few rotor revolutions to a fully developed rotating stall cell. When spike first appears, it is small in circumferential extent and propagates quickly around the annulus, usually between 60 and 80% of the rotor speed. With time, spike grows in amplitude and circumferential extent, and its propagation speed decreases. It is now well established that the fewer blade passages the stall cells occupies, the faster it propagates around the circumference. More details about this short-length-scale disturbance can be found in Hoying [27] and Camp and Day [9].

The second type of stall inception, which was predicted theoretically by Moore and Greetzer [51] before being observed by McDougal et al. [49], is the long-length-scale wavelike disturbance that rotates around the annulus. The term ‘modal waves’ is used to describe this phenomenon because of its wavelike appearance in the velocity traces. An illustration of this pre-stall pattern is given in figure 1.5, which shows the first-order and the second-order mode of this instability. The first-order mode has a wavelength equal to the circumference of the compressor, and the second-order mode has a wavelength of half the circumference. The rotational frequency of modal waves in low-speed axial compressors is usually less than 50% of the rotor speed. Often, modal waves occur only a short time before exceeding the stability limit. In most cases modal waves appear many revolutions before stall and get more intensive as the flow rate

Figure 1.4: Illustration of the short-length-scale disturbance known as spike in an axial compressor - Day [13]
is being reduced. The amplitude of the oscillation usually remains small and seldom exceeds 2 to 3% of the free-stream velocity before flow breakdown occurs. Opposite to spike disturbance, modal wave results in a broad rotating stall cell, which because of its size, rotates comparatively slowly. More details about modal waves can be found in Garnier et al. [26], Day [13], Camp and Day [9] and Hoying [27].

The third aerodynamic instability observed prior to rotating stall can be found already in the stable operating range of compressors. This phenomenon occurs only in compressors with relatively large tip clearance of the rotor blades, usually larger than 4.5% of the blade length. It can be described as a group of superimposed modes or a part-span stall with fluctuating cell numbers. The dominating mode numbers are in the range of half the rotor blade number. This phenomenon is called rotating instability. The propagation speed of rotating instabilities usually varies between 25 and 90% of the rotor speed. Unsteady measurements in the blade tip region of the rotor blades reveal a strong fluctuation of the blade tip vortices. The fluctuating blade tip vortices propagate in circumferential direction along the rotor blade row and form a rotating structure with high mode orders. With respect to the relative frame of reference, the flow within the tip clearance of the rotor can be described by a reversed axial component and a counter-rotating circumferential component of rotating instabilities. As illustrated in figure 1.6, the disturbance propagates as wave-fronts, which means that a spiral propagation of the rotating instability modes against the rotor direction takes place. In figure 1.6 it is shown that spiral propagation is induced by forward flow in the blade tip region and reversed flow near the casing wall. The formation of rotating instabilities occurs near the stability limit, as illustrated in figure 1.7. Reduction of the mass flow rate first results in a gradual increase of the pressure rise up to the maximum value at point e. Then, small amplitude oscillations appear and the performance drops slightly. At point a, an abrupt drop of pressure rise occurs with the appearance of rotating stall. Rotating instabilities are further analyzed in Mailach et al. [45], Mailach [43, 44], Kameier [30], Kameier and Neise [31], Baumgartner et al. [8], Mathioudakis and Breugelmans [48], März et al. [47] and März et al. [46].
1.3. ROTATING STALL

Forward flow in the blade tip region
reversed flow near the casing wall

**Figure 1.6:** Illustration of the pre-stall disturbance known as rotating instabilities, and its way of propagation around the circumference - Mailach [43]

**Figure 1.7:** Schematic drawing of the compressor performance map when rotating instabilities precede rotating stall phenomenon - Mathioudakis and Breugelmans [48]
The rotating stall is axial compressors can be classified as part-span stall or full-span stall, which is based on the size of the stalled region. Usually rotating stall cells block the entire passage between the two blades but not always. In an axial compressor Day [14] has found that in case of the part-span stall, rotating stall cells are formed at the rotor tip or at the stator hub, and that in case of the full-span stall, rotating stall cells are stretched from hub to tip. Since the drop of the pressure rise depends on the amount of disturbed flow in the annulus, the drop of pressure is larger for full-span stall than for part-span stall.

1.3.2 Centrifugal compressors

There is less experimental evidence on rotating stall characteristics in centrifugal compressors than in axial compressors. Here, attempt is made to point out the aerodynamic processes that take place in centrifugal compressors during rotating stall. There seems to be almost no information on pre-stall characteristics or instabilities that occur before rotating stall onset. On the other hand, there is much more known about the interaction between the impeller and diffuser than is known about the interaction between the rotor and stator.

Pre-stall   There are only few reports of phenomena occurring prior to rotating stall in centrifugal compressors. Lawless and Fleeter [36] showed that small-amplitude wave exists before rotating stall occurs, with a warning time of about 14 impeller revolutions. Kang and Kang [33] have measured weak stall precursors upstream of the inducer in a centrifugal compressor with warning times of 0.2 to 2.3 seconds. They found that one-cell, two-cell and three-cells structures of small-amplitude waves grow and decay repeatedly before fully-developed rotating stall occurs. In axial compressors similar warning times are observed when comparing it with the modal waves.

Stall evolution   Based on the compressor performance curve, rotating stall can be progressive or abrupt. In figure 1.8 compressor performance maps are given corresponding to these two types of rotating stall. As shown in figure 1.8, progressive rotating stall has a gradual reduction of total-pressure ratio after stall begins, and abrupt rotating stall has a sudden drop in total-pressure ratio after stall occurs. Abrupt stall usually starts with multiple stall cells while progressive stall always starts with one rotating stall cell. In both cases the number of rotating stall cells may increase as the mass flow rate is being reduced. Typical for abrupt stall is the hysteresis phenomenon, which is also illustrated in figure 1.8. When the mass flow rate is being decreased starting from the stable operating flow condition (point d), the compressor will eventually stall when the stall line (point a) is reached. When increasing the mass flow rate starting from the stalled operating flow condition (point b), a significantly larger mass flow is required for the system to recover (point c) than the mass flow at which rotating stall has occurred (point b). This phenomenon is called hysteresis. Frigne and Braembussche [24] have performed measurements of progressive and abrupt rotating stall showing the differences between them.

Classification   There are various classifications of rotating stall in centrifugal compressors. Based on the compressor component, centrifugal compressor stall can be classified as inducer, impeller or diffuser stall. The inducer stall is usually non-rotating while the stall type in the other two components is rotating stall. Kämmer and Rautenberg [32] have experimentally observed that stall can be local to the inducer and be non-rotating, or it can occupy the full length of the blade channel and be rotating. Diffuser rotating stall can be observed in vaneed as well as vaneless diffusers. Vaned diffuser rotating stall was investigated by Elder and Gill [17], who
1.4. VANELESS DIFFUSER ROTATING STALL

studied the influence of different diffuser variables on the location of the stability limit. Justen et al. [29] investigated the unsteady flow phenomena in the vaned radial diffuser using unsteady pressure measurements on the diffuser front wall and on the suction and pressure surfaces. In many cases, rotating stall in the vaned diffuser is initiated in the semi-vaneless space at the diffuser inlet, but the flow dynamics of this mechanism are not described yet. Also, several observations are made of the vaneless diffuser rotating stall. For example, Abdelhamid and Bertrand [3] and Abdelhamid [2] studied the effects of the vaneless diffuser geometry on rotating stall. Frigne and Braembussche [24] made a distinction between different types of impeller and diffuser rotating stall in a centrifugal compressor with vaneless diffuser. Also Kinoshita and Senoo [34], Dou [15], Shin et al. [57] and Engeda [18, 19, 20] have observed rotating stall by studying the flow phenomena within vaneless radial diffusers.

Based on the stall cell life-time, rotating stall can be classified as mild stall or fully-developed rotating stall. Mild stall is characterized by repeated reappearance and disappearance of rotating stall cells until the fully developed rotating stall occurs and remains. Frigne and Braembussche [24] have observed mild impeller stall prior to the fully-developed rotating stall in their centrifugal compressor configuration. And finally, based on separation location, rotating stall can be classified as blade stall or wall stall.

1.4 Vaneless diffuser rotating stall

Different types of rotating stall in centrifugal compressors exist. This research is concentrated only on the vaneless diffuser rotating stall because of two reasons. First, diffuser rotating stall is the most common rotating stall type in centrifugal compressors. Since the operating range of compressors is of great concern, vaneless diffusers are used more often because they have a wider operating range than the vaned types. Therefore, it is of considerable interest to predict the point of instability inception in the vaneless diffusers. Second, a vaneless diffuser is preferred...
above the vaned diffuser for the study of rotating stall, because of its simple geometry. The non-
rotating diffuser geometry is more preferable for research than the rotating impeller geometry,
looking from an experimental as well as analytical point of view.

**Vaneless diffuser performance** A vaneless diffuser is the simplest and lowest cost diffuser,
but it can easily stall unless it is carefully designed. The advantage of vaned diffusers above
vaneless diffusers, is that diffusion or expansion of air can be carried out in a much shorter flow
path once the air is controlled with vanes.

There are a few vaneless diffuser designs possible. Vaneless diffuser configurations can be
pinched or unpinched. When pinched the abrupt area reduction takes place at the diffuser inlet
to stabilize the flow coming out of the impeller, however some vaneless diffusers are unpinched.
The diffuser walls can be parallel or converging. Parallel wall configuration is often adopted
when convertibility is required for both vaned and vaneless diffuser systems. In this research
the vaneless diffuser is assumed to be unpinched and with parallel diffuser walls.

In the vaneless diffuser no energy is supplied to the air after it leaves the impeller. Then,
neglecting the effect of friction, the angular momentum $v \cdot r$, must be constant. The tangential
velocity component $v$ decreases from the impeller tip to diffuser exit in inverse proportion to
the radius. For a channel of constant diffuser width, the area of flow in the radial direction
is directly proportional to the radius. The radial velocity $u$ will therefore also decrease from
impeller tip to diffuser exit, in accordance with the continuity equation. In the vaneless radial
diffuser the air follows an approximately logarithmic spiral path. For an incompressible fluid
this path can be described by $\tan^{-1}(u/v) = \text{constant}$.

**Literature survey** In the literature different analytical and experimental approaches have
been used to investigate rotating stall in vaneless radial diffusers, and several theories that ex-
plain the vaneless diffuser rotating stall mechanism have been developed. In the early literature
on rotating stall, many researchers have shown that the vaneless diffuser rotating stall originates
as a three-dimensional wall boundary layer instability, but in the later years, studies appeared
showing that the two-dimensional core flow instability might be responsible for rotating stall.
This points towards the possibility that two or may be more flow mechanisms might be responsi-
ble for the occurrence of rotating stall in vaneless radial diffusers. Shin et al. [57] recognized two
different mechanisms for the development of rotating stall in a vaneless diffuser, one dominated
by the extension of the re-entering flow from the diffuser exit and the other dominated by the
growth of a local-flow-separation zone on the hub and shroud side. Measurements of Abdelhamid
and Bertrand [3] also have shown that wide vaneless diffusers behave differently from the narrow
diffusers. This made them presume the existence of more than one set of flow conditions that
could lead to the occurrence of rotating stall. Besides Abdelhamid and Bertrand [3] and Shin
et al. [57], Dou [15] also clearly suggests that the vaneless diffuser performance is different for
narrow and wide diffusers. He showed that distribution of losses in the radial vaneless diffusers
is different for narrow and wide diffusers. These observations imply that a distinction should
be made between narrow and wide vaneless diffusers, as possibly two different flow mechanisms
might lead to the rotating stall instability. In general, one mechanism is associated with the
two-dimensional core-flow instability occurring in wide vaneless diffusers when the critical inlet
flow angle is reached, and the other mechanism is associated with the three-dimensional wall
boundary layer instability occurring in the narrow diffusers.

Although a lot of research is performed on the influence of the vaneless diffuser width on
the unsteady flow phenomena, the exact distinction between the narrow and wide diffusers is
not yet defined. Generally, the distinction between the narrow and wide diffusers is based on
the flow field structure within the diffuser channel. In figure 1.9, the vaneless diffuser flow is schematically represented for narrow and wide vaneless diffuser type. Wide diffusers are assumed to have mainly a two-dimensional core flow in the middle plane of the diffuser, which separates the wall-boundary layers from each other, and narrow diffusers are assumed to have merging wall-boundary layers. In figure 1.9, also the two-dimensional core-flow instability associated with the wide vaneless diffusers is illustrated. Of course, the growth of the wall-boundary layers is dependant on the mass flow rate through the diffuser. The lower the flow angle at the diffuser inlet, the longer the fluid path within the diffuser space, and the larger the wall-boundary layers become. Here, it is assumed that the vaneless diffuser is wide enough to have a fully developed wall-boundary layers and a two-dimensional core-flow in between them.

A three-dimensional approach, where the wall-boundary-layer theory is used to study rotating stall in vaneless diffusers, was applied by Jansen [28], Senoo and Kinoshita [55], Senoo et al. [56], Frigne and van den Braembussche [25] and Dou and Mizuki [16]. They generally state that the effect of the three-dimensional boundary layers near the walls triggers the rotating stall in vaneless diffuser.

On the other hand, Abdelhamid [1], Moore [52] and Tsujimoto et al. [59] have used a two-dimensional approach where the effect of the wall boundary layers is not taken into account. They have used a two-dimensional inviscid and incompressible flow analysis to study the vaneless diffuser rotating stall. These studies suggest the existence of a two-dimensional core flow instability at the onset of rotating stall in vaneless diffusers.

Additionally, measurements of rotating stall in vaneless radial diffusers have been performed by Abdelhamid and Bertrand [3], Abdelhamid [2], Frigne and van den Braembussche [24], Kinoshita and Senoo [34], Dou [15], Shin et al. [57], Engeda [18, 19, 20], Ferrara et al. [23, 21, 22] and Cellai et al. [10, 11]. Experimental work shows a significant influence of the diffuser geometry on the vaneless diffuser performance and stability and on the rotating stall characteristics.

1.5 Research objectives and approach

Little is known about the physics of rotating stall within vaneless radial diffusers of centrifugal compressors. To increase the operating range of centrifugal compressors, a better understanding of rotating stall flow dynamics is required. Therefore, the goals of this research are to analyze the flow dynamics responsible for the occurrence of rotating stall in vaneless diffusers, to study the influence of the vaneless diffuser geometry and operating flow conditions on rotating stall, and to discuss the characteristics and behavior of rotating stall in the vaneless diffuser of centrifugal compressors.

Before performing a fully three-dimensional analysis of the vaneless diffuser flow it is found most preferable to perform a two-dimensional analysis first, to find out if there are two-dimensional flow mechanisms that might contribute to vaneless-diffuser rotating stall. Therefore, rotating stall is investigated assuming that it can be regarded as a two-dimensional instability. If two-dimensional mechanisms exist that contribute to the rotating stall phenomenon, they should occur as a core-flow instability, as illustrated in figure 1.9.

The performed two-dimensional core-flow analysis applies only to wide vaneless diffusers, since there it can be assumed that at compressor operating flow conditions mainly a two-dimensional core flow is present between the diffuser plates. Vaneless diffusers that compare well with this numerical model, as shown later in this report, satisfy \( h/r_2 > 0.1 \), where \( h \) is the diffuser width and \( r_2 \) the diffuser inlet radius. To Analyse the two-dimensional core-flow instability, a two-dimensional numerical flow model of the vaneless diffuser core flow is developed using Computational Fluid Dynamics (CFD). In this model the influence of wall-boundary lay-
This numerical model is verified using a two-dimensional inviscid stability analysis, measurements found in the literature and two experimental setups.

In this thesis, first the two-dimensional numerical model of the vaneless diffuser core flow is described in chapter 2. This numerical model is used to find the instability responsible for the occurrence of rotating stall. Once the two-dimensional rotating instability was found, in chapter 3 experimental validation of the numerical model results is performed. The flow field structures obtained with the numerical model are compared with similar flow field analyses found in the literature, and with the experimentally obtained velocity fields. Performed measurements and corresponding experimental setups are also described in chapter 3. In chapter 4, the physical analysis of the obtained two-dimensional rotating instability is performed. In chapter 5, the influence of the diffuser geometry on the core flow stability and on the characteristics of the two-dimensional rotating instability is investigated. Where possible, data obtained by the current numerical model are compared with the data found in the literature. Finally, in chapter 6 a general discussion and conclusions are given.
Chapter 2

Numerical model

In this chapter, the developed two-dimensional incompressible flow model of the vaneless-diffuser core flow is discussed. The model to be developed is based on a real compressor, in which rotating stall was observed. The reference compressor aerodynamic test facility was situated at Siemens Demag Delaval Turbomachinery B.V. in Hengelo, the Netherlands. The compressor engine details and its operating flow conditions are not reported in this thesis, since they may not be published. The vaneless diffuser of this centrifugal compressor configuration has a diffuser radius ratio \( r_3/r_2 = 1.52 \) and a diffuser width ratio \( h/r_3 = 0.0163 \). First, the scaling of the operating flow conditions and the model geometry is explained. Next, the two-dimensional and numerical aspects of modeling are discussed. At the end, the obtained flow characteristics are described and compared to the rotating stall phenomenon.

2.1 Scaling

The numerical model of the vaneless-diffuser core flow is based on the existing compressor configuration. For security reasons and to make the results of the numerical model comparable to experiments, described in chapter 3, scaling of that compressor into a water model of the centrifugal compressor stage was performed. According to the hydrodynamic analogy, a water model of the same geometry must operate at much lower fluid velocities and impeller speeds than the air compressor configuration. Such a decrease in velocities allows visualization of the unsteady flow phenomena. Hence, the operating conditions of the centrifugal compressor stage near the stability limit have been scaled to obtain a water model that operates near the onset of rotating stall. In this thesis, all results will be, as much as possible, given in dimensionless numbers to make the operating flow conditions generally applicable.

When scaling the compressor flow into a water model, the working medium is being changed from air to water. To keep the operating flow conditions similar, the static pressure coefficient \( \psi \) in the compressor performance map, as shown in figure 3.1, should remain unchanged,

\[
\frac{\Delta p_{st}}{\rho_1} U_2^2 = \text{constant.}
\]

Substitution of the flow parameters and the air and water properties into equation 2.1, gives an approximate value of the impeller tip speed for the near-stall condition in the water stage of the centrifugal compressor, \( U_2 = 5.6 \ [\text{m/s}] \). Once the impeller tip speed of the water configuration is known, the water velocities at the impeller exit can be estimated using the velocity triangle relations. In figure 2.1, a schematic drawing of the impeller outflow is shown, where the inlet flow angle \( \alpha \) and the velocity triangle relations are illustrated. The inlet flow angle \( \alpha \) is defined
as
\[ \alpha = \arctan \left( \frac{u_2}{v_2} \right). \]

According to figure 2.1, for the mean inlet flow angle holds,
\[ \sin \alpha_m = \frac{u_{m2}}{V_2} \quad \text{and} \quad \cos \alpha_m = \frac{U_2 \sigma_0}{V_2}. \]  

where \( u_{m2} \) corresponds to the mean radial velocity at the diffuser inlet and where \( \sigma_0 \) represents the slip correction factor, which is chosen to be constant at 0.9. At near-stall operating flow conditions, the similarity of the velocity triangles has been applied. It is assumed that the mean inlet flow angle \( \alpha_m \) remains unchanged during scaling. Application of the velocity triangles, gives the remaining velocity components.

Furthermore, to scale the diffuser geometry properly, the diffuser radius ratio \( r_3/r_2 = 1.52 \) and the diffuser width ratio \( h/r_3 = 0.0163 \) must also apply to the new diffuser geometry. Here, the two-dimensional vaneless diffuser space is defined with an inlet radius \( r_2 = 0.3227 \) [m]. The corresponding diffuser outlet radius is \( r_3 = 0.4908 \) [m]. The mass flow rate is defined as,
\[ \dot{m} = \rho u_{m2} 2 \pi r_2 h, \]

where \( h \) is the diffuser width, and the impeller speed is defined as
\[ \Omega = \frac{U_2 60}{2 \pi r_2} \text{ [rpm]} \quad \text{and} \quad \omega_i = \frac{2 \pi \Omega}{60} \text{ [rad/s]}. \]

The diffuser geometry, fluid velocities and operating flow conditions that correspond to the modeled vaneless diffuser space, are given in table 2.1.

In order to obtain exact similarity between the compressor flow and the fluid flow within the water model, also the Reynolds number as well as the Mach number must remain unchanged. Although, in chapter 4, it will be shown that the Reynolds similarity is not important for the
### Table 2.1: Geometry and operating flow conditions of the modeled vaneless radial diffuser

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Numerical model reference data</th>
</tr>
</thead>
<tbody>
<tr>
<td>$h$</td>
<td>[m]</td>
<td>0.008</td>
</tr>
<tr>
<td>$r_3$</td>
<td>[m]</td>
<td>0.4908</td>
</tr>
<tr>
<td>$r_2$</td>
<td>[m]</td>
<td>0.3227</td>
</tr>
<tr>
<td>$V_2$</td>
<td>[m/s]</td>
<td>7.66</td>
</tr>
<tr>
<td>$v_2$</td>
<td>[m/s]</td>
<td>7.61</td>
</tr>
<tr>
<td>$u_2$</td>
<td>[m/s]</td>
<td>0.907</td>
</tr>
<tr>
<td>$U_2$</td>
<td>[m/s]</td>
<td>8.455</td>
</tr>
<tr>
<td>$\Omega$</td>
<td>[rpm]</td>
<td>250</td>
</tr>
<tr>
<td>$\omega_i$</td>
<td>[rad/s]</td>
<td>26.2</td>
</tr>
<tr>
<td>$\dot{m}$</td>
<td>[kg/s]</td>
<td>14.7</td>
</tr>
</tbody>
</table>

### Table 2.2: Similarity check of the Reynolds numbers for the water model and compressor operating flow conditions

<table>
<thead>
<tr>
<th>Reynolds number</th>
<th>Compressor configuration</th>
<th>Numerical model</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re_h$</td>
<td>4.8 · 10^4</td>
<td>6.1 · 10^4</td>
</tr>
<tr>
<td>$Re_{r_2}$</td>
<td>1.9 · 10^6</td>
<td>2.5 · 10^6</td>
</tr>
</tbody>
</table>

Core flow instability, the Reynolds number similarity check is performed. To check the flow field similarity, the Reynolds number of the water model and that of the compressor configuration are compared. The Reynolds number based on the diffuser width $h$ is defined as

$$Re_h = \frac{\rho V_2 h}{\mu},$$

and the Reynolds number based on the diffuser inlet radius $r_2$ as

$$Re_{r_2} = \frac{\rho V_2 r_2}{\mu} \equiv Re,$$

where $\mu$ is the dynamic viscosity. By substitution of the fluid properties, the diffuser width or the diffuser inlet radius and the velocity magnitude at the diffuser inlet, both Reynolds numbers can be obtained. This is done for both configurations and the obtained Reynolds numbers are summarized in table 2.2. As shown in table 2.2, the Reynolds numbers are of the same order of magnitude in both cases, which means that the Reynolds number similarity is satisfied. In the remaining, the $Re_{r_2}$ will be used as the characteristic Reynolds number since the influence of the diffuser width is mainly neglected.

It is not possible to satisfy the Mach number similarity when water is used as working medium. In the real compressor configuration the Mach number, defined as $Ma = U_2/c$, is approximately 0.6, which means that the flow is subsonic since $Ma < 1$. In the flow of gases, compressibility effects due to variations in the density may still be neglected as long as the velocity is not too high. According to Kundu [35] the flow may be considered incompressible as long as the free-stream Mach number is smaller than 0.3. For low to medium Mach numbers $0.3 < Ma < 0.8$, density changes can be considerable, but the gas dynamic phenomena like shock waves...
fluid properties are updated, based on the current solution or on the initialized solution

momentum equations are each solved in turn

the pressure correction is derived from the continuity equation and the linearized momentum equations

where appropriate, equations for scalars such as turbulence, energy, species, and radiation are solved

Converged ? STOP

Figure 2.2: Steps of the segregated solution method

...do not have to be considered. The density changes also depend on the geometry. It is assumed that the density changes in this geometry are relatively small and that the compressibility effects are not the main cause cause of the rotating stall instability.

When water is used instead of air one must also take into account the compressibility effects on the velocities occurring in the diffuser. In the air compressor there is a density rise across the diffuser, while in the water model the density remains unchanged. Therefore, in the water model compressibility effects on the velocity should be compensated by increasing the diffuser width towards the diffuser outlet. The radial velocity component in the vaneless diffuser can be defined as follows

\[ u = \frac{\dot{m}}{2 \pi r \rho h}. \]  \hspace{1cm} (2.8)

From equation 2.8 it follows that an increase of \( \rho \cdot h \) in the air diffuser must equal the increase of \( \rho \cdot h \) in the water diffuser channel. In the air diffuser \( \rho \) is the increasing parameter while in the water diffuser \( h \) has to be the increasing parameter. To estimate the magnitude of this effect, the density rise across the diffuser channel in the air compressor configuration is calculated. The air density rise across the vaneless diffuser in the reference air compressor configuration does not exceed 2.5 %, which is found relatively moderate to very small. Therefore, use of parallel diffuser walls in the water diffuser channel is found to be acceptable, and the correction of \( h \) at \( r = r_3 \) with respect to \( h \) at \( r = r_2 \) is not carried out.

2.2 Numerical modeling aspects

To perform the numerical analysis, a commercial software package Fluent was used, which is based on Computational Fluid Dynamics (CFD). Here, the governing integral equations for the conservation of mass and momentum were solved using the finite-volume approach. To solve the governing equations, the segregated solution method is used. Using this approach, the governing equations are solved sequentially, which means segregated from one another. Because the governing equations are non-linear and coupled, several iterations of the solution loop must be performed before a converged solution is obtained. Each iteration consists of the steps illustrated in figure 2.2. These steps are continued until the convergence criteria are met.
2.2. NUMERICAL MODELING ASPECTS

Pressure-velocity coupling In the segregated solver, the Pressure Implicit with Splitting of Operators (PISO) algorithm is used for the pressure-velocity coupling, which is highly recommended for all transient flow calculations. Moreover PISO can be found in Wesseling [61]. One of the limitations of the alternative (SIMPLE or SIMPLEC) algorithms is that new velocities and corresponding fluxes do not satisfy the momentum balance after the pressure-correction equation is solved. As a result, the calculation must be repeated until the balance is satisfied. According to Fluent Users Guide, the PISO algorithm performs two additional corrections to improve the efficiency of the calculations. These are the so-called neighbor correction and skewness correction.

Solver model Although the studied flow is turbulent, the incompressible laminar flow solver is applied, which means that no eddy viscosity but only effects of molecular viscosity are modeled. Use of the turbulence models is omitted because of the production of excessive turbulent dissipation by these models. It is assumed that the instabilities in the two-dimensional core flow have the length scale of the prescribed jet-wake pattern at the diffuser inlet, which corresponds to the large-eddy structures. Since turbulence models capture the diffusion-like character of turbulent mixing, associated with many small eddy structures, they tend to damp the solutions of large structures. To test this assumption, in section 4.6, solutions obtained with a few turbulence models will be compared with the solution obtained using the incompressible viscous flow solver.

Discretization and differencing schemes To obtain the best-accuracy solution for modeling of the incompressible viscous flow, for time differencing a 2nd order implicit time differencing scheme is used, and for discretization of the convection terms the Quadratic Upwind Interpolation for Convective Kinematics (QUICK) scheme, as proposed by Leonard [37], is used. The QUICK scheme is third-order accurate for the spatial discretization of the convection terms and second-order accurate for the diffusion terms.

Initial condition Before starting the numerical simulation, it must be provided with an initial ’guess’ for the solution of the flow field. In first instance, the solution is initialized by setting the values for pressure and velocity at zero. When the desired flow conditions are determined, simulations are initialized by these flow conditions or by the zero pressure and velocity condition, which depends on the situation to be modeled. When the zero pressure and velocity is used
as initial condition, the desired impeller speed is prescribed immediately at the first time step. This means that the impeller startup in this case is a step function.

**Boundary conditions** As shown in figure 2.1, the radial vaneless diffuser with a finite radius ratio $r_3/r_2$ is modeled. Since the core flow of the wide vaneless diffuser is modeled as two-dimensional, and the influence of the wall boundary layers is not taken into account, no diffuser width is modeled. Since the diffuser outlet is connected to the volute, where the flow is highly three-dimensional and most likely swirling, the outlet boundary condition cannot be exactly modeled. Therefore, it is assumed that the volute can act as an open space with constant pressure, and a constant static pressure is prescribed at the diffuser outlet. The impeller outflow at diffuser inlet is represented by the clockwise-rotating jet-wake velocity pattern, as illustrated in figure 2.1. The tangential velocity component $v_2$ is constant around the circumference, and it is related to the impeller tip speed $U_2$ as follows,

$$v_2 = U_2 \sigma_0$$  \hspace{1cm} (2.9)

where $\sigma_0$ is the slip correction factor, which is chosen to be 0.9 in this numerical model, see Cohen [12]. The radial velocity component at the diffuser inlet $u_2$ is described by the periodic hyperbolic tangent function,

$$u_2 = u_{m2} + A \frac{\tanh (DY)}{\tanh (D)}$$,  \hspace{1cm} (2.10)

where $u_{m2}$ is the mean radial velocity, $A$ is the amplitude and $D$ is a constant indicating the steepness of the hyperbolic tangent function. The parameter $Y$ is a sinusoidal function representing the inlet pattern of $N$ number of jet-wakes distributed around a circumference, which is characterized by the angle $\theta$. This jet-wake pattern rotates with the angular impeller speed $\omega_i$. $Y$ is defined as follows:

$$Y = \sin (N \theta + \omega_i t)$$  \hspace{1cm} (2.11)

where $t$ is the current time.

**Reference conditions** At reference condition, 17 impeller blades are modeled, which is same as in the reference compressor configuration. A simplified illustration of the jet-wake prescribed at the diffuser inlet is given in figure 2.3. Proportions of the jet and wake relative to each other are arbitrarily chosen, keeping in mind that the influence of the jet-wake ratios should be investigated. The ratio between circumferential extent of the wake, and circumferential extent of the jet, is called wake-to-jet circumferential extent ratio $\theta_{wake}/\theta_{jet}$, and is arbitrarily chosen to be equal to 1. The ratio between radial intensity of the jet and radial intensity of the wake, is called jet-to-wake radial intensity ratio, and is arbitrarily chosen to be equal to 5.5. Jet-to-wake radial intensity ratio is defined as,

$$\frac{u_{jet}}{u_{wake}} = \frac{u_m + A}{u_m - A}$$  \hspace{1cm} (2.12)

The scaled geometry of the vaneless diffuser space and the corresponding operating flow conditions are determined in order to satisfy the scaling laws described in section 2.1. The diffuser geometry used as a reference condition along with the corresponding velocities and operating flow conditions is given in table 2.1.
2.3. TWO-DIMENSIONAL ROTATING INSTABILITY

**Time step** For the time-periodic calculations, the time step is based on the time scale of the periodicity. For a rotor-stator model, for example, 20 time steps between each blade passing is recommended by the Fluent user’s guide. Since in this case, the modeled number of impeller blades equals 17, the desired amount of time steps per impeller revolution is 340. Then, the time step \( \Delta t \) depends on the impeller tip speed \( U_2 \) according to the following relation,

\[
\Delta t \leq \frac{2 \pi r_2}{340 U_2}
\]  

(2.13)

which means, the higher the impeller tip speed, the smaller the required time step. At the scaled operating flow condition, \( U_2 = 8.455 \text{ [m/s]} \), the recommended time step is \( \Delta t \leq 0.00071 \text{ [s]} \). To show the importance of a proper time-step choice, two simulations are performed, one with the time step as recommended by equation 2.13, and the other with a larger time step than recommended. The solutions obtained with \( \Delta t = 0.001 \text{ [s]} \) and \( \Delta t = 0.0006 \text{ [s]} \) are shown in figure 2.4, where solutions are represented by contours of velocity magnitude. This figure shows that the solution can drastically change if the time step is too large, and that a proper choice of the time step at all times is very important. The time step should also not be too small, because then computations will become unnecessarily time consuming. In the following computations the time step is always chosen to satisfy equation 2.13.

**Mesh size** To mesh the two-dimensional vaneless-diffuser geometry, a simple two-dimensional quadrilateral grid is used. The number of grid points in \( r \) and \( \theta \) direction is chosen such that the aspect ratio of the grid cells is close to 1. To achieve a smooth development of the diffuser flow in time, condition \( t_{cell} < \Delta t \) must be satisfied. Here, \( \Delta t_{cell} \) is the time necessary for a particle traveling with a certain speed to cross the distance of the grid cell \( \Delta r \) or \( \Delta \theta \). The choice of the grid size is based on this condition and is applied to the reference operating flow conditions. For a given impeller tip speed and the corresponding time step, the required size of the grid cells can be determined with the following relation,

\[
\Delta r \approx \Delta \theta \approx U_2 \Delta t_{cell}
\]  

(2.14)

For the impeller tip speed \( U_2 = 8.455 \text{ [m/s]} \) and the corresponding maximum allowed time step \( \Delta t = 0.00071 \text{ [s]} \), a mesh seeding of \( 62 \times 750 \) satisfies above condition. To check the solution, also a simulations with doubled mesh seeding of \( 124 \times 1500 \) cells are performed. Since refinement of the mesh did not result in change of solution, a mesh of \( 62 \times 750 \) grid points is applied to the numerical model.

**Convergence and stability** Convergence can be hindered by a number of factors such as, large numbers of computational cells, overly conservative under-relaxation factors, and complex flow physics, and therefore, it must be monitored. All performed unsteady simulations satisfy the default convergence criterion of Fluent, which requires that the continuity, \( x \)-velocity and \( y \)-velocity residual decrease to \( 10^{-3} \) for all equations. This convergence criterion is applied at each time step.

### 2.3 Two-dimensional rotating instability

After varying the tangential and mean radial velocity components at the diffuser inlet, two different flow conditions are obtained, which are shown in figure 2.5. Moreover the two operating flow conditions and the transition between them can be found in Ljevar et al. [39]. The solutions
in figure 2.5 are represented by contours of velocity magnitude, where the red color represents high velocity and the blue color represents low velocity regions. The solution on the left represents the stable operating flow condition and the solution on the right represents the unstable operating flow condition.

At stable operating flow condition, outward pointed and reversed flow regions seem to alternate near the diffuser outlet. They seem to be part of the counter-clockwise vortices, as indicated in figure 2.5, which are held near the diffuser outlet by the outlet boundary condition. The number of these regions exactly corresponds to the number of the prescribed jet-wakes. During the unstable operating flow condition, seven counter-clockwise rotating vortex structures are propagating around the circumference, which is associated with the rotating stall phenomenon. This structure represents the two-dimensional rotating instability. These two solutions were obtained by varying the tangential and radial velocity component, which means that they occur under different operating flow conditions.

By gradually decreasing the mean radial velocity component in time, keeping the impeller speed unchanged, a gradual transition from the stable operating flow condition into the two-dimensional rotating instability can be generated, as illustrated in figure 2.6. Here, successive solutions are shown as a function of time, along with the corresponding mean inlet flow angle, which decreases as the mean radial flow component is being decreased. The solutions in figure 2.6 are represented by the contours of velocity magnitude. At a certain point, the periodic structure of the stable operating flow condition falls apart, and after a few impeller revolutions, a new operating flow condition is obtained.

The two-dimensional rotating instability seems to occur due to the interaction between the jet-wake flow near the diffuser inlet and the alternating flow pattern near the diffuser outlet. At the stable operating flow condition, one jet-wake pair at the diffuser inlet corresponds to one clockwise and counter-clockwise vortex pair near the diffuser outlet. When the mean inlet-flow angle is large, the jet-wake pattern flows mainly towards the alternating flow pattern, and they gear perfectly into each other. When the mean inlet flow angle decreases, the flow direction of the jet-wake pattern becomes more circumferential. As soon as the mean jet-wake flow is circumferential enough to pass underneath the alternating pattern instead of interacting with it, alternating flow areas become unequal in size and the instability starts to occur. When $\alpha_m$ is decreased further the pattern falls apart as illustrated in figure 2.6. From this point, smaller and weaker vortices seem to merge with the larger ones, which makes the larger vortices to grow bigger. Finally, a number of these rotating cells reaches their final size and propagation speed, and they distribute equally around the circumference. This way of transition from the stable operating flow condition into the two-dimensional rotating instability indicates that the instability originates near the diffuser outlet and that it probably depends on the interaction between the diffuser inlet and outlet flow conditions. More about the origin of this instability will be discussed in chapter 4.

To check if the non-periodicity of the grid cells with respect to the number of impeller blades could be responsible for this two-dimensional rotating instability, the mesh size is varied once more. Here, the number of grid cells around the circumference is chosen to be exact the multiple of the number of impeller blades $N$. Since in this case $N$ is chosen to be 17, simulations for the reference geometry were repeated with the mesh existing of $765 \times 62$ cells. The same solutions were obtained for the same operating flow conditions as obtained earlier with the mesh of $750 \times 62$ cells, which indicates that the instability is not caused by the mesh non-periodicity in circumferential direction.
2.3. TWO-DIMENSIONAL ROTATING INSTABILITY

Figure 2.4: Solutions obtained at two different time steps, left $\Delta t = 0.001$ [s] and right $\Delta t = 0.0006$ [s], which are represented by contours of velocity magnitude; $r_3/r_2 = 1.52$ and $\alpha_m > 11^\circ$ stable unstable $\alpha_m = 11^\circ$ $\alpha_m = 7.4^\circ$ $\alpha_m = 7.2^\circ$ $\alpha_m = 6.3^\circ$ $\alpha_m = 5.9^\circ$ $\alpha_m = 5.4^\circ$ $\alpha_m = 3.9^\circ$

Figure 2.5: On the left stable and on the right unstable operating flow condition for $r_3/r_2 = 1.52$; both solutions are represented by contours of velocity magnitude $\alpha_m = 11^\circ$ $\alpha_m = 7.4^\circ$ $\alpha_m = 7.2^\circ$ $\alpha_m = 6.3^\circ$ $\alpha_m = 5.9^\circ$ $\alpha_m = 5.4^\circ$ $\alpha_m = 3.9^\circ$

Figure 2.6: Transition from stable to unstable operating flow condition for $r_3/r_2 = 1.52$; the mean radial velocity is decreased while the impeller speed remains unchanged.

Figure 2.4: Solutions obtained at two different time steps, left $\Delta t = 0.001$ [s] and right $\Delta t = 0.0006$ [s], which are represented by contours of velocity magnitude; $r_3/r_2 = 1.52$ and $\alpha_m > 11^\circ$ stable unstable $\alpha_m = 11^\circ$ $\alpha_m = 7.4^\circ$ $\alpha_m = 7.2^\circ$ $\alpha_m = 6.3^\circ$ $\alpha_m = 5.9^\circ$ $\alpha_m = 5.4^\circ$ $\alpha_m = 3.9^\circ$

Figure 2.5: On the left stable and on the right unstable operating flow condition for $r_3/r_2 = 1.52$; both solutions are represented by contours of velocity magnitude $\alpha_m = 11^\circ$ $\alpha_m = 7.4^\circ$ $\alpha_m = 7.2^\circ$ $\alpha_m = 6.3^\circ$ $\alpha_m = 5.9^\circ$ $\alpha_m = 5.4^\circ$ $\alpha_m = 3.9^\circ$

Figure 2.6: Transition from stable to unstable operating flow condition for $r_3/r_2 = 1.52$; the mean radial velocity is decreased while the impeller speed remains unchanged.
Figure 2.7: Solutions obtained at different operating flow conditions for $r_3/r_2 = 1.52$ and $t = 0.6 \text{ s}$, showing that there is a stability limit when $\alpha_c \approx 10.5^\circ$. 
2.4 Stability limit

By varying the impeller tip speed and the mean radial velocity, it is found that the two-dimensional rotating instability occurs when the mean flow angle $\alpha_m$ becomes small. The core flow is considered unstable when periodicity of the stable operating flow condition, as shown in figure 2.5, is disturbed. Solutions obtained at different operating flow conditions are given in figure 2.7, where the solutions are represented by contours of velocity magnitude. All these conditions are obtained when the impeller startup is characterized by a step function. Different impeller speeds are represented by the Reynolds number, which is based on the corresponding impeller tip speed $U_2$ and the diffuser inlet radius $r_2$. For each impeller tip speed, critical inlet flow angle $\alpha_{cr}$ near the stability limit is determined, and it is found that $\alpha_{cr} \approx 10.5^\circ$. The two-dimensional rotating instability occurs when the mean flow angle at the diffuser inlet becomes smaller than the critical inlet flow angle, $\alpha_m < \alpha_{cr}$. This criterion is in good agreement with the observations found in the literature, where often the same stability criterion is used, because rotating stall is often found to occur when the flow angle at the diffuser inlet becomes small.

The corresponding performance curves of the vaneless diffuser flow are given for different impeller speeds in figure 2.8, where the normalized static pressure rise $\psi$, is plotted against the flow coefficient $\phi$, both defined in section 1.2. The flow coefficient $\phi$ is based on the diffuser width ratio $h/r_2 = 0.15$, which equals the ratio in the radial flow pump experimental setup and which satisfies $h/r_2 > 0.1$. Figure 2.8 shows an abrupt pressure drop on the left side of the stability limit along with the occurrence of the two-dimensional rotating instability, which is one of the characteristics of the rotating stall phenomenon. On the right side of the stability limit, a stable operating flow is obtained.

At the right side of the stability limit where a stable operating flow is found, the pressure rise
across the diffuser very slightly decreases with decreasing mass flow rate. Usually, in vaneless diffusers that operate at stable operating flow conditions, the pressure rise slightly increases with decreasing mass flow rate. The drop of the pressure rise in the numerical model might be due to the fact that no wall-boundary layer effects are taken into account. In practice, shear layers near the walls create resistance, which has to be overcome. At lower mass flow rates, the resistance as well as the pressure losses decrease, which results in a higher pressure rise over the diffuser at low mass flow rates than at high mass flow rates.

2.5 Discussion

The obtained two-dimensional rotating instability is similar to rotating stall in the sense that it fully develops within a few impeller revolutions, and that it consists of a number of rotating cells that propagate around the circumference with a fraction of the impeller speed. For this particular diffuser configuration it is found that 7 or 8 rotating cells occur, which propagate with approximately 40% of the impeller speed. Besides that, the obtained performance map in figure 2.8 shows a sudden drop in pressure coefficient when the instability occurs. This is similar to the performance curves associated with rotating stall, shown in figures 1.7 and 1.8. To check the similarity of this two-dimensional rotating instability with the rotating stall phenomenon, an experimental validation will be performed in chapter 3. Furthermore, the results obtained with the two-dimensional numerical model will be compared with the experimental data found in the literature in chapter 5.
Chapter 3

Experimental validation

Although pressure transducer or hot-wire measurements can offer interesting information about the unsteady flow phenomena, the information gained only from discrete measurement points does not reveal the overall image of the rotating stall mechanism. Therefore, the focus is set on studying the rotating stall flow mechanism by analyzing the local velocity obtained with PIV and pressure data obtained with microphone measurements. To check the validity of the current numerical model, these data are compared with the two-dimensional linear stability analysis found in the literature, and with the experimental data obtained with two different experimental setups. One experimental setup is based on a no-through-flow water model and the other on an open radial flow pump stage. The comparison is based on a number of parameters, that are characteristic for the rotating stall instability. These are the mean and the critical flow angle at the diffuser inlet $\alpha_m$ and $\alpha_{cr}$, the number of cells $m$ and the relative propagation speed of the rotating cells $\omega_{rs}/\omega_i$, where $\omega_{rs}$ and $\omega_i$ are the angular velocities of the rotating cells and the impeller respectively.

The studied operating flow conditions corresponding to each of the two experimental setups and to the two-dimensional numerical model are schematically illustrated in a performance map given in figure 3.1. To compare the numerical model results with the experimental results, the geometry and operating flow conditions in the numerical model are adjusted to the experiments. In the upper row the operating range is given for the two-dimensional numerical model as scaled in chapter 2. In the middle, on the left, the operating flow conditions of the experimental setup corresponding to the radial flow pump stage are given. On the right, the operating flow conditions of the numerical model are given, which is used for comparison with this setup. In the bottom row, on the left, water model operating flow conditions are given, and on the right those of the numerical model used for comparison. The corresponding Reynolds number for each model is reported in this figure and will not be repeated below. According to figure 3.1 the Reynolds numbers similarity is achieved between the compared experimental and numerical simulations. The unstable operating flow condition corresponds in all cases to the region on the left side of the stall line shown in figure 3.1, which means that $\alpha_m < \alpha_{cr}$. Since in all cases the unstable operating flow condition is obtained just after the stability limit, they are comparable. The pressure and velocity fields found in the literature are not obtained at the same operating flow conditions as the numerical model results. Therefore, in this case, the comparison is only qualitative. Although the operating flow conditions are not exactly the same, similarities between different models are obtained. There are also a few differences, which are explained as much as possible.

This chapter is divided in five sections. First, the velocity and pressure fields are discussed, which are obtained with the two-dimensional numerical model. Then, qualitative comparison
between the numerical model and the velocity and pressure field found in the literature is performed. Next the two experimental setups and the corresponding results are discussed. In the final section, the overall comparison is made between the current numerical model and the obtained experimental data and the data found in the literature.

### 3.1 Operating flow conditions

In this section, the two operating flow conditions obtained by the two-dimensional numerical model are analyzed. This is done by discussing the velocity and pressure fields corresponding to the stable and unstable operating flow condition. To better recognize the flow structures within the velocity field, besides the velocity, also the velocity fluctuations are analyzed. Velocity fluctuations are obtained by subtracting the mean velocity field, averaged along the circumference at each radius, from the instantaneous velocity field.

The velocities, the velocity fluctuations and the pressure contours corresponding to the stable operating flow condition are shown in figure 3.2. The velocities show a wavelike pattern near the diffuser outlet. In the velocity fluctuation field this pattern near the diffuser outlet consists of clockwise and counterclockwise rotating areas. This pattern is a result of the prescribed jet-wake velocity pattern at the diffuser inlet, which generates these structures. The constant pressure condition at the diffuser outlet keeps the pattern within the diffuser space near the outlet. In the velocity fluctuation field in figure 3.2, the prescribed jet-wake pattern at the diffuser inlet can be clearly distinguished from the alternating flow pattern near the diffuser outlet. Here, the prescribed jet-wake pattern is better visible than in the velocity field since the mean flow field is subtracted, and the jet and wakes can be clearly distinguished from each other. The pressure contours are also given in figure 3.2. The counter-clockwise rotating structures propagating near the diffuser outlet correspond to the low pressure regions.

The velocities, the velocity fluctuations and the pressure contours corresponding to the twodimensional rotating instability are shown in figure 3.3. The velocity field as well as the velocity fluctuation field clearly show the counter-clockwise rotation of the flow field within the rotating cells and the clockwise rotation of the flow field in between the rotating cells. The counter-clockwise rotation of the cells is a consequence of the clockwise rotation of the jet-wake pattern.

---

**Figure 3.1:** Operating range of the current numerical model and the experimental setups used for experimental validation.
prescribed at the diffuser inlet, which acts on the diffuser flow as a large vortex. The vortex structures or rotating cells are kept within the diffuser space by the constant pressure condition at the diffuser outlet. The pressure field obtained by the two-dimensional core-flow model, which is also given in figure 3.6, shows that rotating stall cells correspond to the low pressure regions and reversed flow between the rotating cells corresponds to the high pressure regions within the pressure field.

3.2 Comparison with the data found in the literature

In this section, the velocity and pressure fields, obtained with the two-dimensional numerical model, are compared with those measured by Tsurusaki et al. [60] and Nagashima and Itoh [53], and with those obtained by the two-dimensional inviscid stability analysis performed by Tsujimoto et al. [59].

Tsurusaki et al. [60] have performed rotating stall measurements in the vaneless diffuser space with \( r_3/r_2 = 2.35 \) and \( h/r_2 = 0.1 \). This results in the streamlines based on measured fluctuating velocity components, which are shown on the left-hand side of figure 3.4. In figure 3.4, the impeller rotation direction is indicated by the arrow near the impeller rotation speed sign \( \Omega \). Since the impeller rotates in counter-clockwise direction, the streamlines suggest that two clockwise rotating stall cells are present at points \( G_1 \) and \( G_2 \), with two corresponding counter-clockwise rotating vortices at points \( G_3 \) and \( G_4 \). In the same figure, the velocity fluctuations are given, which are obtained with the two-dimensional numerical model for the same diffuser radius ratio. For this diffuser radius ratio four rotating cells are obtained with the two-dimensional numerical model, as shown in figure 3.4. Also here, the counter-clockwise rotation direction of the impeller results in clockwise rotating cells and counter-clockwise rotating regions between the rotating cells. Therefore, the rotation of the flow structures corresponds well to those measured by Tsurusaki et al. [60]. Despite the difference in the cell number, the two velocity fields look similar to each other. The same flow structures are observed in both cases in the same arrangement. Since the number of rotating cells is not only dependent on the diffuser geometry but also on the mass flow rate through the diffuser space, the number of rotating cells is not fixed for a given diffuser radius ratio and can vary with the mass flow rate. Therefore, the number of rotating cells is not being compared, but only their arrangement and the flow field structure.

Tsujimoto et al. [59] have compared velocity and pressure fluctuation fields resulting from their two-dimensional inviscid stability analysis with their own experiments for \( r_3/r_2 = 2 \), and with the experiments of Nagashima and Itoh [53] for \( r_3/r_2 = 3 \). The velocity and pressure fluctuation fields from Tsujimoto et al. [59] are compared with the velocity and pressure fluctuation fields obtained with the current numerical model. The mean flow angle is in both cases smaller than the critical flow angle, \( \alpha_m < \alpha_{cr} \), which means that both operating conditions correspond to the region left of the stall line, shown in figure 3.1. The velocity and pressure fluctuation fields that correspond to \( r_3/r_2 = 2 \) are given in figure 3.5, and those that correspond to \( r_3/r_2 = 3 \) are given in figure 3.6.

In figure 3.5, the impeller rotation direction is indicated by the arrow. In the measured and calculated flow fields by Tsujimoto et al. [59], two rotating stall cells are represented, while in our numerical simulation four rotating cells are shown. For the diffuser radius ratio \( r_3/r_2 = 2 \), the number of rotating cells obtained just past the stability limit is in the current numerical model \( m = 4 \) and in the two-dimensional inviscid stability analysis \( m = 5 \). This is in good agreement as will be shown in chapter 5. In figure 3.5, the number of cells \( m = 2 \) is probably modeled because this number of cells is obtained experimentally. The fact that the number of cells in
Figure 3.2: Flow field corresponding to the stable operating flow condition for $r_3/r_2 = 1.52$ at $\alpha_m = 11^\circ$: a) velocity, b) velocity fluctuation and c) pressure contours
3.2. COMPARISON WITH THE DATA FOUND IN THE LITERATURE

Figure 3.3: Flow field corresponding to the two-dimensional rotating instability for \( r_3/r_2 = 1.52 \) at \( \alpha_m = 6.8^\circ, m = 7 \): a) velocity, b) velocity fluctuation and c) pressure contours
Figure 3.4: Comparison for $r_3/r_2 = 2.35$ between a) velocity streamlines based on the fluctuating component from an experiment performed by Tsurusaki et al. [60], $m = 2$ and b) velocity fluctuation field from the current numerical model, $\alpha_m = 11.4^\circ$, $\omega_{rs}/\omega_i = 0.21$, $m = 4$.

Figure 3.5: Upper half: velocity fluctuation fields, and lower half: pressure contours, for $r_3/r_2 = 2$ a) two-dimensional inviscid stability analysis of Tsujimoto et al. [59], $\alpha_m = 13.8^\circ$, $\omega_{rs}/\omega_i = 0.088$, $m = 2$ b) experiment performed by Tsujimoto et al. [59], $\alpha_m = 3.3^\circ$, $\omega_{rs}/\omega_i = 0.24$, $m = 2$ and c) current numerical model, $\alpha_m = 10.5^\circ$, $\omega_{rs}/\omega_i = 0.25$, $m = 4$. 
the experimental case is lower than that obtained by the two models, is probably due to the additional dissipation effects caused by the turbulence and the wall boundary layers. But despite the difference in the presented number of rotating cells, similarity of the flow fields can again be noticed. In all three cases, the pressure field consists of the alternating high and low pressure regions, and the velocity fluctuation field consists of the clockwise and counter-clockwise rotating areas. Since the rotating cells in the numerical model have the opposite rotation direction than the impeller, it is also considered to be the case in the other two situations. Then, the rotating stall cells correspond to the low pressure regions in the pressure fluctuation fields, which is also the case in the current numerical model. Figure 3.5 shows that similar flow fields are observed between the results obtained by Tsujimoto et al. [59] and those obtained by the current numerical model.

Since in Tsujimoto et al. [59] only the pressure fluctuation fields were given for $r_3/r_2 = 3$, in figure 3.6 only pressure fluctuation fields can be compared. Also here, the impeller rotation direction is indicated by the arrow. In the calculated pressure fluctuation fields by Tsujimoto et al. [59], and measured pressure fluctuation fields by Nagashima and Itoh [53], two rotating stall cells are found, while in the current numerical simulation three rotating cells are shown. The reason for this difference is the same as in the previous figure. But again, despite the difference in the presented number of rotating cells similarity of the flow fields is observed.

### 3.3 Water model

Since the two-dimensional core-flow instability was obtained by the numerical model, an experimental verification of this instability was desired. To analyze the two-dimensional rotating instability, a water model of the vaneless diffuser space was built. For visualization of the flow field a Particle Image Velocimetry (PIV) is used, as described in Raffel et al. [54]. To obtain a water model that operates near the stability limit, near stall operating flow conditions of an existing air compressor were scaled to water operating flow conditions, as described in section 2.1.

To make a proper comparison with the numerical model, the core flow between the vaneless diffuser walls needs to be two-dimensional. Therefore, also the diffuser width condition $h/r_2 >
Table 3.1: Scaled geometry and operating flow conditions of the water model of the wide vaneless diffuser

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Water experimental setup</th>
</tr>
</thead>
<tbody>
<tr>
<td>( h )</td>
<td>[m]</td>
<td>0.0033</td>
</tr>
<tr>
<td>( r_3 )</td>
<td>[m]</td>
<td>0.2</td>
</tr>
<tr>
<td>( r_2 )</td>
<td>[m]</td>
<td>0.13</td>
</tr>
<tr>
<td>( V_2 )</td>
<td>[m/s]</td>
<td>4.86</td>
</tr>
<tr>
<td>( v_2 )</td>
<td>[m/s]</td>
<td>4.83</td>
</tr>
<tr>
<td>( u_2 )</td>
<td>[m/s]</td>
<td>0.58</td>
</tr>
<tr>
<td>( U_2 )</td>
<td>[m/s]</td>
<td>5.37</td>
</tr>
<tr>
<td>( \Omega )</td>
<td>[rpm]</td>
<td>394</td>
</tr>
<tr>
<td>( \omega_i )</td>
<td>[rad/s]</td>
<td>41.3</td>
</tr>
<tr>
<td>( \dot{m} )</td>
<td>[kg/s]</td>
<td>1.6</td>
</tr>
</tbody>
</table>

0.1, needs to be satisfied. In the previous chapter it is shown that the two-dimensional rotating instability occurs when the flow angle at the diffuser inlet is small enough, and in the following chapter it is shown that the impeller startup needs to be rapid enough to obtain the proper operating flow conditions. To capture the two-dimensional rotating instability, the water model must satisfy all these conditions.

### 3.3.1 Experimental setup

A water model of the vaneless radial diffuser was built in an already available cylindrical tank with given geometry. Therefore, the diffuser geometry and its operating flow conditions were based on the geometry of this cylindrical tank. The experimental setup consists of two concentric cylinders, where a moving fluid is confined between the two cylinders. A schematic drawing of the experimental setup is given in figure 3.7. The inner cylinder represents an impeller body, it rotates in the clockwise rotation direction and it has five flat blades mounted perpendicularly to it. The outer cylinder is non-rotating, and it is a determining factor for the radial extent of the vaneless diffuser, which is placed between the two concentric cylinders. Since the available cylindrical tank with the radius of \( r_4 = 0.32 \) [m] was used as non-rotating outer cylinder, the diffuser outlet radius is chosen to be \( r_3 = 0.2 \) [m] to leave some space between the diffuser exit and the cylinder wall. According to the scaling laws given in section 2.1, the corresponding diffuser inlet radius, diffuser width and impeller tip speed are then \( r_2 = 0.13 \) [m], \( h = 0.0033 \) [m] and \( U_2 = 5.37 \) [m/s], respectively. Knowing the diffuser geometry, the corresponding velocities and operating flow conditions of the experimental setup are also calculated and reported in table 3.1.

The diffuser walls are placed at the mid-height of the impeller body. The lower diffuser wall is fixed to the outer cylinder, while the upper diffuser wall is floating on the water surface. The diffuser width was varied by changing the water level in the cylindrical tank. Although the upper diffuser wall is floating on the water surface, its rotational degree of freedom was fixed, which prevented the upper diffuser wall to move with the moving fluid. To enable optical access to the measurement area from the top side, the upper diffuser wall was made of transparent material. Since the optical access from the side wall was also desired, the outer cylinder is surrounded by the square tank that is also filled with water in order to prevent optical deformation by the refractive index of the cylindrical surface. The outer cylinder as well as the surrounding
Figure 3.7: Cross section of the water model experimental setup, top view on the left-hand side and side view on the right-hand side
rectangular tank were both made of transparent material.

As shown in figure 3.7, the laser beam enters the measurement area through the side walls. On top of the setup, two high-speed (CCD) cameras are placed to make images of the illuminated measurement plane. The overlapping views of the two cameras and the impeller rotation direction are shown in figure 3.8.

To allow comparison with the numerical model, the vaneless diffuser space is made to be wide. The diffuser width is set at \( h = 0.03 \) [m], which results in a satisfactory diffuser width ratio of \( h/r_2 = 0.23 \). To enable rapid impeller startup, an axis connects the impeller body directly to the electric motor situated on top of the setup. To generate the two-dimensional rotating instability the fluid between the diffuser walls is brought into movement only by rotating the impeller body. Since there is no through-flow, the mean flow angle at the diffuser inlet is approximately zero, which is a satisfactory condition to generate the two-dimensional rotating instability.

### 3.3.2 Measurements

To visualize the vaneless diffuser flow using the PIV technique, polyamid particles of 50 [\( \mu \text{m} \)] were used for seeding. These particles were neutrally buoyant and small enough to follow the fluid closely. During the measurement the particles are illuminated by the laser sheet, which is spread in the desired measurement plane. The laser light is generated by a pulsed Nd-YAG laser. Synchronous with the laser pulse frequency, the cameras make images of the illuminated particles. Correlation of the successive camera images results in the two-dimensional velocity field of the measurement plane.

Because the speed of rotation of the impeller body could not be varied, the PIV measurements were performed at only one operating flow condition corresponding to an impeller speed of \( U_2 = 0.5 \) [m/s]. This impeller speed is lower than scaled in table 3.1. The Reynolds number decreases by a factor of the order of 10, but the flow continues to be of a turbulent character.
The measurements were started when $U_2 = 0$ [m/s], and stopped when the fully developed velocity field was obtained. In the first few seconds of the measurement the impeller accelerates, which is shown in figure 3.9 where the traveled distance of one impeller blade is plotted versus time.

To measure the flow field, PIV measurements were performed in the $r-\theta$ plane, which is parallel to the diffuser walls. The measurement plane was generated at the mid-height of the vaneless diffuser space, which is associated with the core flow region.

To investigate if the core flow between the diffuser plates is two-dimensional, and to study the influence of the secondary flows on the core flow region, PIV measurements in the vertical plane were also performed. The vertical measurement plane was perpendicular to the $r-\theta$ plane, and it intersected the rotation axis of the impeller body.

To check the reproducibility, the PIV measurements in both measurement planes were repeated several times, and each time the same instabilities and flow field development were observed.

### 3.3.3 Measurement results

The measurement results from the $r-\theta$ plane are shown in figure 3.10, where the left images correspond to camera view 1 and the right images to camera view 2, as indicated in figure 3.8. The measured flow fields at three different time instants are shown, namely $t = 1.6$ [s], $t = 2.2$ [s] and $t = 2.9$ [s]. The vectors in figure 3.10 represent the instantaneous velocity field while the colored background represents the vorticity. The horizontal and vertical scales indicate the distance in millimeters.

As can be seen in figure 3.10, the rotating vortex structures that propagate circumferentially
are found to exist. Two overlapping camera views were sufficient to capture two successive vortex structures.

Measurements in the $r-\theta$ plane show that in the early stage of the measurement, a separation line is formed between the high velocity region near the rotating impeller body and the low velocity region near the wall. As can be seen in figure 3.10, counter-clockwise vortex structures develop on the separation line between the two flow regions. Because the impeller startup is measured, the radius of the separation line moves towards the diffuser outlet and eventually it leaves the camera views. During the first 6 seconds of the measurement, the moving separation line and the occurring vortex structures are captured by the two camera views. After $t > 6 \text{[s]}$, the camera views capture only the high velocity region where the tangential velocity component gradually decreases towards the outlet, and where no vortex structures are observed.

The observed vortex structures have a relative diameter of $d_s/r_2 \approx 0.31$ and propagation speed $\omega_{rs}/\omega_i = 0.15$. The radial distance between the two successive vortices is estimated at $l_s/r_2 \approx 1$, and the relative vorticity magnitude of the rotating cells is $\zeta r_2/U_2 \approx 1.8$. The estimated uncertainties in $d_s/r_2$, $\omega_{rs}/\omega_i$, $l_s/r_2$ and $\zeta r_2/U_2$ are $\pm 0.02 [-]$, $\pm 0.02 [-]$, $\pm 0.1 [-]$ and $\pm 0.2 [-]$ respectively.

### 3.3.4 Comparison with the numerical model

For comparison, a numerical simulation is performed where similar conditions are used as in the measurements. The diffuser inlet radius corresponds to the radius of the rotating impeller body. The outlet boundary condition was changed to the no-slip wall boundary condition at the radius corresponding to that of the outer cylinder. The same impeller tip speed is used. At the diffuser inlet the jet-wake profile consisting of 5 jet-wakes is prescribed, which corresponds to the five blades mounted on the impeller body. The mean radial velocity equal to zero is applied, which corresponds to no through flow situation. The impeller speed corresponds to the Reynolds number $Re = 6.5 \cdot 10^5$, which is the same as in the water model as shown in figure 3.1. The obtained numerical results are shown in figure 3.11 for $t = 0.15 \text{[s]}$, $t = 0.6 \text{[s]}$, $t = 1.05 \text{[s]}$, $t = 1.8 \text{[s]}$, $t = 3.9 \text{[s]}$ and $t = 6 \text{[s]}$ since the start of impeller rotation. The upper solutions are represented by the velocity magnitude while the lower solutions are represented by the vorticity magnitude.

The flow structures obtained by the numerical model, which are shown in figure 3.11, are similar to the experimentally observed structures. In both cases, clockwise rotation of the impeller body results in counter-clockwise rotating vortices at the separation line where the velocity gradient is high. The solutions of vorticity magnitude clearly show that the final vortex structures are generated at the shear layer between the high and low velocity region. First, a wavelike disturbance at the separation line occurs, which later grows and develops into the fully developed vortices.

The vortex structures obtained with the numerical simulation have a relative diameter $d_s/r_2 = 0.35$ and a propagation speed $\omega_{rs}/\omega_i = 0.07$. The relative distance between two successive vortices is $l_s/r_2 \approx 1$. The relative vorticity magnitude of the rotating cells is $\zeta r_2/U_2 \approx 2.5$. Compared to the experimental case, good agreement is found between the size of the rotating cells and the distance between them. A difference exists between the propagation speed and the vorticity magnitude. The vorticity magnitude in the experimental case is smaller than that obtained with the numerical model. It is assumed that this difference is caused by the nearby dissipation effects due to the secondary flows which are present in the experimental case and are not modeled in the numerical case. The influence of the secondary flows in the experimental case is discussed in the next section. The propagation speed in the experimental case is approx-
Figure 3.10: PIV images in the $r - \theta$ plane, with on the left camera view 1 and on the right camera view 2; $r_3/r_2 \approx 1.5$, $\alpha_m = 0^\circ$, $m \approx 7$, $\omega_{rs}/\omega_i = 0.15$; colors represent vorticity.
Figure 3.11: Numerical solutions corresponding to the configuration of the water model, where the upper row represents contours of velocity magnitude and the bottom row represents contours of vorticity; $r_3/r_2 \approx 1.5$, $\alpha_m = 0^\circ$, $m = 10$, $\omega_{rs}/\omega_i = 0.07$
3.3. WATER MODEL

Figure 3.12: Radial distance of the rotating cell from the impeller as a function of time

approximately twice the speed obtained by the numerical model. This difference is probably due to the different radial distance of the cells from the impeller. To compare the radial distance of the cells from the impeller, in figure 3.12 this distance is plotted versus time for several measurements and for the corresponding numerical simulation. Figure 3.12 shows that the distance of the cells from the impeller is approximately two times larger in the numerical case than in the experimental case. This explains the factor two difference in propagation speed, which is inversely proportional to the radial distance of the cells from the impeller, as shown in the next chapter.

Not only the magnitude of the radial distance from the impeller, but also its trend versus time is different. In the experimental case, the distance from the impeller has a wavelike trend, which is not the case in the numerical simulation. This difference in radial distance from the impeller and its wavelike trend is probably due to the different impeller representation, which can result in a slightly different diffuser inlet profile and different slip effects near the impeller body. In the experimental case a rotating impeller body with five thin blades is used, while in the numerical case a rotating jet-wake velocity profile is prescribed at the diffuser inlet where the circumferential length of the jets and the wakes is equal to each other. The difference in radial distance of the cells from the impeller might also be due to the influence of the wall boundary layers. The presence of the shear forces near the walls in the experimental case might cause the rotating cells to move slower towards the diffuser outlet. This will be discussed in chapter 5.

3.3.5 Secondary flows

The velocity fields measured in the vertical measurement plane are shown in figures 3.13 and 3.14. In figure 3.13, PIV images are shown that are made at $t = 0$ [s], $t = 1/15$ [s], $t = 2/15$ [s], $t = 4/15$ [s], $t = 7/15$ [s] and $t = 9/15$ [s], while figure 3.14 continues with the PIV images at
Figure 3.13: Instantaneous velocity field at different time instants measured in the vertical plane of the wide vaneless diffuser of the water model, for $t = 0 \, [s]$ to $t = 9/15 \, [s]$, $r_3/r_2 \approx 1.5$, $\alpha_m = 0^\circ$
Figure 3.14: Instantaneous velocity field at different time instants measured in the vertical plane of the wide vaneless diffuser of the water model, for $t = 13/15$ [s] to $t = 31/15$ [s], $r_3/r_2 \approx 1.5$, $\alpha_m = 0^\circ$
$t = 13/15 \text{[s]}, \ t = 1 \text{[s]}, \ t = 19/15 \text{[s]}, \ t = 23/15 \text{[s]}, \ t = 25/15 \text{[s]} \text{ and } t = 31/15 \text{[s]}$. In the first image in figure 3.13, corresponding to $t = 0 \text{[s]}$, the intrusion of the impeller blade into the measurement plane is indicated by the rectangle on the left. At $t = 0 \text{[s]}$ the impeller rotation is started. Depending on the position of the impeller blades, which generated the jet-wake effect, the flow field is pulled towards the impeller or pushed away from the impeller. Immediately after the impeller rotation is started the three-dimensional flow structures are observed close to the impeller. In the beginning, $t < 1 \text{[s]}$, the flow is mainly two-dimensional, while a small region near the impeller is three-dimensional. After $t > 1\text{[s]}$, the three-dimensional region becomes larger and it progresses further towards the diffuser outlet until the entire diffuser channel consists of highly three-dimensional flow structures.

Figures 3.13 and 3.14 show that the secondary flows do exist, but they do not cover the entire flow region. The secondary flows are clearly present in the high-velocity flow region close to the impeller body, while the remaining part of the flow is two-dimensional. As shown in figure 3.10 the rotating cells are generated near the separation line between the high-velocity region and the low-velocity region. Since these structures lie mainly in the low-velocity region, which is associated with the two-dimensional flow region in figure 3.13, it is presumed that they are mainly two-dimensional.

### 3.4 Vaneless diffuser measurements

The water model described in the preceding section was good enough to check the physical nature of the results obtained with the two-dimensional numerical model, but it could not be used to describe the rotating stall phenomenon as it occurs in real vaneless diffusers. Therefore, an existing experimental setup at the Laboratoire de Mecanique de Lille (ENSAM, Lille, France) was used, which is described in Wuibaut et al. [63]. For flow visualization, again PIV is used. Simultaneously with the PIV, pressure trace measurements were performed to determine the number of rotating stall cells and their propagation speed.

#### 3.4.1 Experimental setup

The experimental setup consists of an open loop radial flow pump stage with air as working medium. A photograph and schematic drawing of the experimental setup are given in figure 3.15. As can be seen in figure 3.15 no volute is used. The diffuser exit is connected to the open space with constant pressure.

The top view of the experimental setup is illustrated in figure 3.16. The radial pump impeller consists of 7 impeller blades, which make a blade angle of $22.5^\circ$ with the peripheral direction. The outlet part of the impeller has a two-dimensional design, meaning that blades are parallel to the axial direction. The impeller outlet radius and width are $r_{2i} = 0.2566 \text{[m]}$ and $h_i = 0.0385 \text{[m]}$, respectively. The vaneless diffuser space has an inlet radius equal to $r_2 = 0.2571 \text{[m]}$. Since it was designed to have a diffuser radius ratio of 1.52, its outlet radius is equal to $r_3 = 0.390 \text{[m]}$. The diffuser width is $h = 0.039 \text{[m]}$, which results in $h/r_2 = 0.15$. With this $h/r_2$ the diffuser width condition $h/r_2 > 0.1$ is satisfied.

As shown in figure 3.15, CCD cameras were positioned on the shroud side of the diffuser to make PIV images. To enable optical access to the vaneless diffuser flow from the shroud side, the upper diffuser wall was made of transparent material. In figure 3.17, the cross-section A-A, as indicated in figure 3.16, is illustrated. As shown in figure 3.17, the laser sheet enters the vaneless diffuser space through the diffuser outlet. To cover the radial extent of the vaneless diffuser space, two cameras were used. As can be seen in figure 3.16, camera views slightly
Figure 3.15: On top a photograph, and on the bottom a schematic drawing of the radial flow pump stage
overlap near the middle radius of the vaneless diffuser space. One camera captures the impeller exit and the inlet of the vaneless diffuser space, while the other camera captures the diffuser exit area and the region just outside of the diffuser.

To perform pressure measurements, six microphone holes were drilled at the upper diffuser wall whose positions are indicated in figure 3.16. Three radial positions correspond to $r_{p1} = 0.297$ [m], $r_{p2} = 0.3335$ [m] and $r_{p3} = 0.370$ [m], and the angular distance between each pair of microphones is $\Delta \theta_s = 75^\circ$.

### 3.4.2 Pressure measurements

To determine the performance map of the stage and to identify the unsteady flow phenomena at different mass flow rates, pressure measurements were performed. Therefore, Bruel & Kjaer condenser microphones of type 4135 were mounted flush on the upper diffuser wall at six positions as indicated in figure 3.16.

Pressure measurements were performed at different mass flow rates for two impeller speeds, corresponding to $\Omega = 1200$ [rpm] and $\Omega = 1800$ [rpm]. For each impeller speed, pressure measurements were performed at discrete values of the flow coefficient $\phi$, as defined in equation 1.1, starting at $\phi = 0.0074$ and increased in 16 steps up to $\phi = 0.0272$.

The identification of rotating stall structures was performed using auto-power and cross-power spectra of the pressure signals between two microphones at the same radius. The cell number $m$ and the propagation speed of the observed rotating stall cells $\omega_{rs}/\omega_i$ were determined by the following two relations,

$$\omega_p = m \omega_{rs}, \quad (3.1)$$

and

$$\Delta \varphi = m \theta_s, \quad (3.2)$$

Here, $\theta_s$ is the angular distance between the two sensors, $\omega_{rs}$ is the rotating stall angular speed, $\omega_p$ is the angular speed of the pressure signal and $\Delta \varphi$ the phase shift between the two sensors. $\omega_{rs}$ and $\Delta \varphi$ are both determined using the cross-power spectra. Since the number of cells is an integer, calculated phase shift in equation 3.2 should be a multiple of the angular distance between the sensors in the case of rotating stall. Otherwise, it may not be rotating stall or not a fully developed rotating stall.

The obtained performance characteristics are summarized in figure 3.18 for both impeller speeds of rotation. In the upper plot the static pressure coefficient and in the lower plot the number and the propagation speed of the stall cells are plotted versus the relative flow rate.

As shown in figure 3.18, a very good similarity has been observed between the two impeller speeds. Furthermore, figure 3.18 shows that the unstable operating flow condition is divided in three zones with well defined number of rotating cells and two zones where the number of rotating cells is not so well defined, which are called transition zones. In the transition zones less stable rotating structures are existent, which separate the zones with 3, 2 and 4 rotating stall cells. The mean flow angle $\alpha_m$ corresponding to the stability limit is estimated at approximately $15^\circ$, with an estimated accuracy of $\pm 0.5^\circ$.

The pressure signals at three different radii were well synchronized and gave the same results. Therefore, pressure measurements at only one radius were sufficient to study the characteristics of rotating stall, when performed simultaneously with the PIV measurements. Since the impeller speed did not influence the rotating stall instability, PIV measurements were performed at only one impeller speed of rotation, $\Omega = 1680$ [rpm]. To visualize the flow structures at different zones, as indicated in figure 3.18, PIV measurements were performed at six different flow rates,
3.4. VANELESS DIFFUSER MEASUREMENTS

Figure 3.16: Schematic drawing of the experimental setup top-view

Figure 3.17: Schematic drawing of the A-A cross-section
Figure 3.18: Microphone measurements at different relative mass flow rates for two impeller speeds corresponding to $Re = 6.2 \cdot 10^5$ and $Re = 9.2 \cdot 10^5$; up the performance characteristics and down rotating stall characteristics, $r_3/r_2 = 1.52$
3.4. VANELESS DIFFUSER MEASUREMENTS

The PIV measurements were performed at mid-height of the vaneless diffuser space, as shown in figure 3.17. Therefore, a laser sheet generated by the Nd-Yag pulsed laser was used to illuminate the seeding, which was the smoke of burning incense. From the shroud side, two CCD cameras synchronized with the laser pulses were used to make the images of the illuminated seeding particles.

At each flow rate 13 measurement sequences were performed. During each measurement sequence 80 successive image pairs were recorded, which are then analyzed using image cross-correlation to obtain the instantaneous velocity fields. Simultaneously with the PIV measurements, pressure signals were measured with two microphones mounted at $r_{p1} = 0.297 \text{ [m]}$.

3.4.3 Flow field measurements

The pressure transducer measurements performed simultaneously with the PIV measurements indicated exactly the same instability characteristics as obtained previously. At $\phi = 0.0074$ rotating stall with $m = 3$ and $\omega_{rs}/\omega_i = 0.28$ is found, at $\phi = 0.0156$ rotating stall with $m = 2$ and $\omega_{rs}/\omega_i = 0.27$ is found, and at $\phi = 0.0212$ rotating stall with $m = 4$ and $\omega_{rs}/\omega_i = 0.18$ is found. Here, the estimated accuracy of the relative propagation speed $\omega_{rs}/\omega_i$ is $\pm 0.01 \text{ [-]}$.

The well defined rotating stall zones were separated by the transition zones at $\phi = 0.0133$ and $\phi = 0.0181$. At $\phi = 0.0252$ stable operating flow condition was observed.

The PIV recording frequency was 4 [Hz], which means that every seven impeller revolutions one successive image pair is recorded. This frequency was not fast enough to follow the propagation of the rotating stall cells with respect to one impeller blade, but it was good enough to capture the rotating stall cells at random positions with respect to the impeller, and to study their size and position within the vaneless diffuser space.

To illustrate what parts of the rotating stall cells are captured by the two cameras, in figure 3.19 a schematic drawing is given, showing the path of the rotating stall cell with respect to the joined camera views. At all flow rates except for $\phi = 0.0252$, rotating stall cells were observed. In figure 3.20 instantaneous flow fields are shown, where only a part of the rotating stall cells is captured. Since the size of the rotating stall cells is larger than that of the camera views, only the front, middle, or back of the passing cell is captured by the cameras. Since the PIV frequency is too low to make successive images of the passing rotating stall cells through the joined camera views, in figure 3.20 randomly captured images are ordered such to illustrate the passing stall cell. All images shown in figure 3.20 correspond to the measurements of the
same relative flow rate, which is in this case \( \phi = 0.0074 \).

As illustrated in the schematic drawing in figure 3.19 only the lower part of the rotating stall cells is captured by the cameras, while the upper part of the cells lies outside of the vaneless diffuser. In figure 3.20, the first image shows the front of the rotating stall cell entering the camera view, next image represents the middle of the rotating stall cell, and images 3 and 4 represent the back of the stall cell. When the separate PIV velocity charts from figure 3.20 are joined together one rotating stall cell can be reconstructed as shown in figure 3.21. The diffuser inlet and diffuser outlet are indicated by imaginary lines. The observed rotating stall cells move from left to right through the joined camera views. From figures 3.20 and 3.21 can be deduced that the center of the rotating stall cells lies outside of the vaneless diffuser. The relative size of the rotating stall cells is estimated to be \( d_s/r_2 \approx 1 \), with estimated accuracy of \( \pm 0.1 \) [-].

The results shown in figure 3.20 are also representative for the other five flow rates where rotating stall is observed. Irrespective of the flow rate, the same flow structures were observed having approximately the same size. Only their radial distance from the impeller seemed to increase with increasing mass flow rate. This observation explains the decreasing trend of their propagation speed with increasing flow rate, which is obtained by the pressure measurements. The further away from the impeller, the lower the propagation speed of the rotating stall cells seems to be.

During the stable operating flow condition, obtained at \( \phi = 0.0252 \), no unsteady flow phenomena were observed. The captured vaneless diffuser flow moves smoothly from the lower left corner to the upper right corner of the images, as shown in figure 3.22, where two images corresponding to the stable operating flow condition are given.

For comparison with the experimental results, numerical model simulations with conditions close to those of the experimental setup are performed. The same diffuser radius ratio of \( r_3/r_2 = 1.52 \) is used. At the diffuser inlet 7 rotating jet-wakes are prescribed, which corresponds to 7 impeller blades. To indicate the differences and similarities between the jet-wake profiles at the diffuser inlet, the velocity profiles as measured or prescribed at the impeller exit are given in figure 3.23. The jet-wake profiles in both cases correspond to the same mean flow angle, \( \alpha_m \approx 2.9 \, ^\circ \). From figure 3.23 follows that the velocity profiles differ in structure. In the numerical case the tangential velocity is constant while the radial velocity varies. In the experimental case the tangential velocity component varies, while the radial velocity component is approximately constant. Therefore, also the \( \tan \alpha_2 \) profile is different, which is defined as \( \tan \alpha_2 = u_2/v_2 \). Despite the local differences, it is assumed that both jet-wake patterns should have nearly the same effect on the vaneless diffuser core flow as long as the mean flow angle \( \alpha_m \) at the diffuser inlet does not differ. In chapter 4 it is shown that the jet-wake shape and intensity have negligible influence on the two-dimensional rotating instability in the numerical case. In addition, it can be interesting to also numerically investigate the influence of the measured jet-wake pattern on the vaneless diffuser core flow.

The velocity field obtained with the numerical model at the prescribed mean flow angle \( \alpha_m = 10^\circ \) is given in figure 3.24. Here, the vectors represent the velocity. This operating flow condition represents the two-dimensional rotating instability with 7 rotating cells that propagate around the circumference with \( \omega_{rs}/\omega_i = 0.4 \).

Comparison of the flow fields from figures 3.20 and 3.22 with those in figure 3.24, shows that similar flow structures are observed. The clockwise rotating impeller generates in both cases the counter-clockwise rotating stall cells. While experimentally 2, 3 and 4 rotating stall cells were observed, in the numerical model the most unstable mode consists of 7 rotating stall cells. The relative size of the cells is nearly the same in both cases, which is \( d_s/r_2 \approx 1 \). Also a quite good agreement is found in the relative vorticity, which is of the same order in both cases.
Figure 3.20: Instantaneous flow fields corresponding to the rotating stall condition at $\phi = 0.0074$, ordered such to reconstruct the passing rotating stall cell through the camera views, $r_3/r_2 = 1.52$, $m = 3$, $\omega_{rs}/\omega_1 = 0.28$
Figure 3.21: Joined PIV velocity charts reconstructing a single rotating stall cell measured in the wide vaneless diffuser behind the radial flow pump impeller, \( r_3/r_2 = 1.52, m = 3, \omega_{rs}/\omega_i = 0.28 \)

Figure 3.22: Instantaneous flow fields corresponding to the stable operating flow condition at \( \phi = 0.0252 \)
Figure 3.23: Comparison between the experimentally measured velocity profile at the impeller exit (upper graph) and the numerically prescribed velocity profile at the diffuser inlet (lower graph) for $\alpha_m \approx 2.9 \, ^\circ$ and $r_3/r_2 = 1.52$
Figure 3.24: Numerical model solution at $\phi = 0.026$, $r_3/r_2 = 1.52$, $\alpha = 10^\circ$, $m = 7$, $\omega_r s/\omega_i = 0.4$, which is obtained with the experimental operating flow conditions.

Figure 3.25: Numerical model solution at $\phi = 0.039$, $r_3/r_2 = 2$, $\alpha = 15^\circ$, $m = 5$, $\omega_r s/\omega_i = 0.24$, which is obtained with the experimental operating flow conditions.
relative vorticity of the cells obtained with the numerical model is $\zeta_r/\omega_2 \approx 5$, while the relative vorticity of the measured rotating stall cells is $\zeta_r/\omega_2 \approx 1$. The slight difference in vorticity is most likely due to the additional dissipation effects in the experimental case, where the turbulent viscosity and the wall-boundary layers are present. The lower propagation speed of the cells in the experimental case, $\omega_i/\omega_i = 0.15 - 0.27$, with respect to the numerical model, $\omega_i/\omega_i = 0.4$, is probably due to the larger distance of the cells from the impeller. In the previous chapter it is shown that the propagation speed is influenced by the distance of the cells from the impeller.

The relative distance of the cells from the impeller in the experimental case is $r_s/r_2 = 1.6$, while in the numerical model this is $r_s/r_2 = 1.35$. Since in the experimental case the diffuser outlet is not so clearly delimited as in the numerical case, the rotating stall cells incline to keep a slightly larger distance from the impeller than in the numerical model. Therefore, the measured rotating stall characteristics better compare to the numerical simulation with $r_3/r_2 = 2$ instead of $r_3/r_2 = 1.52$. The slightly larger diffuser radius ratio allows the cells to take more distance from the impeller, which is also the case in the experimental setup. The numerical solution of the two-dimensional rotating instability obtained for $r_3/r_2 = 2$ is shown in figure 3.25. Here, the number of rotating cells is found to equal 5 and their relative size has slightly increased to $d_i/r_2 \approx 1.3$. The propagation speed of the cells is $\omega_i/\omega_i = 0.24$, the relative distance of the cells from the impeller is $r_s/r_2 = 1.77$, and the relative vorticity is $\zeta_r/\omega_2 \approx 2.1$. Generally, a better agreement is found between the experimental results and the numerical model with $r_3/r_2 = 2$. The remaining difference in the number of rotating stall cells is probably due to the wall friction and/or the turbulence effects, which are not included into the numerical model. Influence of the turbulence effects is studied in the previous chapter. In the next chapter, it is shown that the wall-boundary layers also promote dissipation of the flow structures, which can result in the lower number of rotating cells. The number of cells in the numerical model is also found to decrease with decreasing mass flow rate. Since the numerical operating flow condition at $\phi = 0.039$, shown in figure 3.25, is obtained just after the stability limit is exceeded and since the mean flow angle is of the same order of magnitude, the numerical model with $r_3/r_2 = 2$ can be compared to the experimental case with $m = 4$ at $\phi = 0.0212$. In the numerical as well as in the experimental case the number of cells is lower for lower mass flow rates.

## 3.5 Discussion

In this section the overview of the compared experimental and numerical conditions is given, which is accompanied by the final discussion about each comparison. The estimated uncertainty of the critical flow angle $\alpha_{cr}$ and the propagation speed ratio $\omega_i/\omega_i$ in the numerical calculations is approximately $\pm 0.1$ [°] and $\pm 0.02$ [−] respectively.

**Literature** The overview of the qualitative comparison of the current numerical model with the data found in the literature is given in table 3.2. In the first column, all models involved into the comparison are given. In the second, third and fourth column it is shown which models are compared for diffuser radius ratios $r_3/r_2 = 2$, $r_3/r_2 = 2.35$ and $r_3/r_2 = 3$ respectively. The results corresponding to diffuser radius ratios $r_3/r_2 = 2$, $r_3/r_2 = 2.35$ and $r_3/r_2 = 3$ are given in figures 3.5, 3.4 and 3.6. For each model it is indicated if the pressure or velocity field was compared, and if available the rotating stall characteristics are given. Figures 3.4 and 3.5 show that in all cases, the velocity fluctuation field consists of the clockwise and counter-clockwise rotating areas, and figures 3.5 and 3.6 show that the pressure fields consist of the alternating high and low pressure regions. It is assumed that the rotating stall cells correspond to the
low pressure regions in the pressure fluctuation fields, since this is obtained with the current numerical model. Figures 3.4 to 3.6 show that similar flow fields are observed between the pressure and velocity fields found in the literature and those obtained by the current numerical model.

**Water model**  The overview of the quantitative comparison of the current numerical model with the experimental data, obtained with the water model of wide vaneless diffuser, is given in table 3.3. Figures 3.10 and 3.11 have shown that quite good agreement is found when comparing the observed flow structures. But the data overview in table 3.3 shows that there are also slight differences between the characteristics of the observed structures. There is a difference in radial distance of the cells from the impeller. This difference might be due to the different impeller representations, which can result in a slightly different diffuser inlet profile and different slip effects near the impeller body. But, it might also be due to the influence of the wall boundary layers, which might cause the rotating cells to move slower towards the diffuser outlet. The number of cells in the experimental case is estimated to be lower than the obtained number of cells in the numerical case. This difference might be due to the dissipation effects generated by the nearby secondary flows and wall-boundary layers, which are not taken into account in the current numerical model. But it might also be due to the different characteristic length of the separation line at which the cells start to develop. The difference in propagation speed is most likely due to the different radial distance of the cells from the impeller. In the next chapter it is shown that the propagation speed of the cells is most dependent on the distance of the cells from the impeller. Furthermore, the vorticity magnitude in the experimental case is a little bit smaller than that obtained with the numerical model. It is assumed that this difference is caused by the nearby dissipation effects due to the secondary flows which are present in the

<table>
<thead>
<tr>
<th>Model</th>
<th>$r_3/r_2 = 2$ compared in figure 3.5</th>
<th>$r_3/r_2 = 2.35$ compared in figure 3.4</th>
<th>$r_3/r_2 = 3$ compared in figure 3.6</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tsurusaki et al. [60] experimental data</td>
<td>velocity &amp; pressure $\alpha_m = 13.8^\circ$ $m = 2$ $\omega_{rs}/\omega_i = 0.088$</td>
<td>velocity $m = 2$</td>
<td>pressure $m = 2$</td>
</tr>
<tr>
<td>Tsujimoto et al. [59] analytical data</td>
<td>velocity &amp; pressure $\alpha_m = 3.3^\circ$ $m = 2$ $\omega_{rs}/\omega_i = 0.24$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tsujimoto et al. [59] experimental data</td>
<td>velocity &amp; pressure $\alpha_m = 3.3^\circ$ $m = 2$ $\omega_{rs}/\omega_i = 0.24$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nagashima and Itoh [53] experimental data</td>
<td></td>
<td></td>
<td>pressure $m = 2$</td>
</tr>
<tr>
<td>current numerical model</td>
<td>velocity &amp; pressure $\alpha_m = 10.5^\circ$ $m = 4$ $\omega_{rs}/\omega_i = 0.25$</td>
<td>velocity $\alpha_m = 11.4^\circ$ $m = 4$ $\omega_{rs}/\omega_i = 0.21$</td>
<td>pressure $\alpha_m = 29.1^\circ$ $m = 3$ $\omega_{rs}/\omega_i = 0.17$</td>
</tr>
</tbody>
</table>

Table 3.2: Overview of the comparison with the velocity and pressure fields found in the literature
3.5. DISCUSSION

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Water model</th>
<th>Numerical model</th>
</tr>
</thead>
<tbody>
<tr>
<td>$r_3/r_2$</td>
<td>$\approx 1.54$</td>
<td>$\approx 1.54$</td>
</tr>
<tr>
<td>$h/r_2$</td>
<td>0.23</td>
<td>$\infty$</td>
</tr>
<tr>
<td>$\phi$</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>$m(l_s/r_2)$</td>
<td>$\approx 7 (\approx 0.6)$</td>
<td>$10 (\approx 0.72)$</td>
</tr>
<tr>
<td>$\omega_{rs}/\omega_i$</td>
<td>$\approx 0.15$</td>
<td>0.07</td>
</tr>
<tr>
<td>$d_s/r_2$</td>
<td>$\approx 0.31$</td>
<td>0.35</td>
</tr>
<tr>
<td>$r_s/r_2$</td>
<td>$\approx 0.62$</td>
<td>$\approx 1.15$</td>
</tr>
<tr>
<td>$\zeta r_2/U_2$</td>
<td>$\approx 1.8$</td>
<td>$\approx 2.5$</td>
</tr>
</tbody>
</table>

**Table 3.3:** Overview of the comparison with the water model of the wide vaneless diffuser

The overview of the quantitative comparison of the current numerical model with the experimental data, obtained in the wide vaneless diffuser behind the radial flow pump impeller, is given in table 3.4. Figures 3.20, 3.21 and 3.24 show that quite good agreement is found in the flow structures. But data in table 3.4 show that the experimental data better agree with the numerical model with $r_3/r_2 = 2$ than $r_3/r_2 = 1.52$. This difference is probably due to the different effect of the outlet boundary condition. The distance of the outlet experimental case and are not modeled in the numerical case.

Resemblance between the structures measured in the $r - \theta$ plane and structures obtained with the numerical model shown in figure 3.11, shows that the obtained numerical solution has a physical character. This experiment has helped to demonstrate that the obtained two-dimensional rotating instability is physical and not numerical.

During the experiment the impeller startup condition is investigated, where shear-layer instability occurs at the separation line between the high-velocity and the low-velocity region. This shear-layer instability results in a growing wavelike disturbance, which eventually deforms and generates a row of rotating vortex structures. The vortex structures will represent the stable operating flow condition or the two-dimensional rotating instability depending on the flow angle prescribed at the diffuser inlet. Since in this experiment no through-flow is present, the mean radial velocity component and thus also the mean flow angle at the diffuser inlet equal zero. Because the two-dimensional rotating instability occurs when the mean inlet flow angle is small, it is assumed that the vortex structures obtained during the startup represent the vortex structures of the two-dimensional rotating instability.

Looking at the early development of rotating cells obtained with the current numerical model, as shown in figure 3.11 for the experimental case, it is most likely that the shear-layer instability at the separation line between the high velocity and the low velocity fluid initiates a number of vortical structures. In both cases, the growing wave transforms into the vortex structures that propagate around the circumference with a fraction of the impeller speed. Because of the different environment in which this instability is generated, further development, characteristics and behavior of the rotating cells is different in the case of the experiment from that of the real vaneless diffuser. The presence of the outer cylinder wall along with no through-flow condition contributes to the dissipation of the rotating cells, which is not the case in the vaneless diffuser environment. The influence of the no-slip wall boundary condition at the diffuser outlet is discussed in more detail in section 4.5.

**Vaneless diffuser** The overview of the quantitative comparison of the current numerical model with the experimental data, obtained in the wide vaneless diffuser behind the radial flow pump impeller, is given in table 3.4. Figures 3.20, 3.21 and 3.24 show that quite good agreement is found in the flow structures. But data in table 3.4 show that the experimental data better agree with the numerical model with $r_3/r_2 = 2$ than $r_3/r_2 = 1.52$. This difference is probably due to the different effect of the outlet boundary condition. The distance of the outlet
boundary condition is in the experimental setup further away than the actual diffuser outlet, and the effect of the outlet boundary condition is probably smaller than in the numerical case. In chapter 5, influence of the diffuser radius ratio is studied. It is found that the critical flow angle increases with increasing diffuser radius ratio. The further the alternating flow pattern near the outlet is from the impeller, the larger the flow angle needs to be in order to have a gearing jet-wake pattern into the alternating flow pattern near the diffuser outlet. Since the diffuser space increases with increasing diffuser radius ratio, the rotating cells become larger and the number of rotating cells decreases in order to fit into the circumferential length of the diffuser. Generally, a good agreement is found between the experimental results at $\phi = 0.0212$ and the numerical model with $r_3/r_2 = 2$. The remaining difference in the number of rotating stall cells is most likely due to the wall friction and/or the turbulence effects, which are not included into the numerical model. Furthermore, there is a difference in the vorticity magnitude of the rotating cells. This difference might also be due to the additional dissipation effects in the experimental case, where the turbulent viscosity and the wall boundary layers are present.

The experimental setup is found suitable for comparison with the two-dimensional numerical model because it meets the model in several conditions. The same diffuser radius ratio is used of $r_3/r_2 = 1.52$. The vaneless diffuser with the diffuser width ratio $h/r_2 = 0.15$ can be considered as a wide vaneless diffuser since the condition $h/r_2 > 0.1$ is satisfied. Therefore, the assumption can be made that a mainly two-dimensional core flow is existent around the mid-height of the diffuser. At the impeller exit, the impeller blades are characterized by the two-dimensional design, which contributes to the two-dimensional character of the vaneless diffuser core flow. The Mach number corresponding to the desired impeller speed during the measurements, $Ma = U_2/c \approx 0.13$, is small enough to assume that the flow is incompressible, which corresponds to the incompressible numerical model. And in the end, no volute is used, which meets the constant pressure condition prescribed at the diffuser outlet in the numerical model.

An added value of the experimental setup is its significance as an intermediate configuration between the two-dimensional numerical model and the real air compressor configuration. It has many two-dimensional aspects as mentioned above, but on the other hand it has also many additional effects that can be found in the real air compressor configurations. The working medium is air, the wall boundary layers near the diffuser walls are present, the effect of turbulence

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Vaneless diffuser</th>
<th>Numerical model $r_3/r_2 = 1.52$</th>
<th>Numerical model $r_3/r_2 = 2.0$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$r_3/r_2$</td>
<td>1.52</td>
<td>1.52</td>
<td>2</td>
</tr>
<tr>
<td>$h/r_2$</td>
<td>0.15</td>
<td>$\infty$</td>
<td>$\infty$</td>
</tr>
<tr>
<td>$\alpha_{cr}$</td>
<td>$\approx 15^\circ$</td>
<td>$\approx 11^\circ$</td>
<td>$\approx 17^\circ$</td>
</tr>
<tr>
<td>$\phi$</td>
<td>0.0074</td>
<td>0.0156</td>
<td>0.0212</td>
</tr>
<tr>
<td>$\alpha_m$</td>
<td>3$^\circ$</td>
<td>7$^\circ$</td>
<td>11$^\circ$</td>
</tr>
<tr>
<td>$m$</td>
<td>3</td>
<td>2</td>
<td>4</td>
</tr>
<tr>
<td>$\omega_{rs}/\omega_i$</td>
<td>0.28</td>
<td>0.27</td>
<td>0.18</td>
</tr>
<tr>
<td>$d_s/r_2$</td>
<td>$\approx 1$</td>
<td>$\approx 1$</td>
<td>$\approx 1$</td>
</tr>
<tr>
<td>$r_s/r_2$</td>
<td>$\approx 1.6$</td>
<td>$\approx 1.7$</td>
<td>$\approx 1.8$</td>
</tr>
<tr>
<td>$\zeta r_2/U_2$</td>
<td>$\approx 1$</td>
<td>$\approx 1$</td>
<td>$\approx 1$</td>
</tr>
</tbody>
</table>

Table 3.4: Overview of the comparison with the PIV measurements performed in the wide vaneless diffuser behind the radial flow pump impeller.
is present, the real impeller is used, and the interaction between the impeller and diffuser flow is possible.

Despite the differences between the experimental setup and the numerical model, the obtained instabilities and flow structures look qualitatively much alike. The agreement of the above experimental results with the numerical model results confirms the existence of the mainly two-dimensional rotating instability in the semi-realistic vaneless radial diffuser environment, and supports the hypothesis that in the real vaneless radial diffusers the similar instability exists. In the next chapters, the validated numerical tool will be used to analyse some physical and geometric parameters.
Chapter 4

Instability analysis

In this chapter, the physical characteristics of the obtained two-dimensional core-flow instability are investigated. As possible origins of the instability, the jet-wake interaction at the inlet, the tangential acceleration and the shear-layer instability of the layer between the impeller region and the outflow region are investigated. To that end, the obtained two-dimensional rotating instability is compared to the two-dimensional inviscid stability analysis found in the literature. Next, the impeller startup is investigated. Then, the influence of the jet-wake pattern prescribed at the diffuser inlet is analyzed by varying a number of jet-wake parameters. Then, the outflow condition is changed to further investigate the origin of the instability. Finally, the damping characteristics of the increased viscosity and engineering turbulence models are investigated.

4.1 Two-dimensional inviscid stability analysis

An analytical model of the two-dimensional vaneless diffuser core-flow model was developed by Tsujimoto et al. [59]. They have investigated vaneless diffuser rotating stall based on a linear two-dimensional inviscid flow analysis, and have also found that the vaneless diffuser flow has a two-dimensional, inviscid and rotational flow instability. Similar analyses to that of Tsujimoto et al. [59] have been performed by Abdelhamid [1] and Moore [52], also showing that a two-dimensional instability of the vaneless diffuser space is possible.

Tsujimoto et al. [59] studied rotating stall from the point of view that it can be regarded as a two-dimensional inviscid flow instability. They assumed that the flow is inviscid and incompressible and that the disturbance is small enough to allow a linear analysis. It is assumed that the steady flow in the vaneless diffuser space is represented by a uniform radial rotating flow. In order to determine the unsteady flow components in the vaneless diffuser due to vorticity, the vorticity transport equation is linearized and solved. It is also assumed that the relative flow exits the impeller tangentially to the vanes. Since the obtained unsteady flow components due to vorticity, did not satisfy this boundary condition, two potential flow components are added to the velocity field induced by the vorticity. Then, the unsteady flow components are obtained, which satisfy the boundary condition at the diffuser inlet. Therefore, the unsteady flow field in the vaneless diffuser space is represented by the velocity induced by vorticity and by the two additional potential flow components. At the diffuser outlet, a constant pressure is prescribed. Here, the simplified case is considered where the impeller effects are not taken into account. In the simplified case, the following boundary conditions are considered: the unsteady flow components at the diffuser inlet are equal to zero and the unsteady pressure at the diffuser outlet is equal to zero. By applying all boundary conditions and assumptions, a condition for existence
of a non-trivial solution is obtained. Once the diffuser radius ratio \( r_3/r_2 \) and the number of cells \( m \) is given, the flow angle \( \alpha \) and the propagation speed \( \omega_r \) can be determined. More details about the two-dimensional inviscid flow model are given in appendix A.

The two-dimensional inviscid stability analysis meets the two-dimensional numerical model in the sense that they are both two-dimensional, incompressible and have a constant pressure condition prescribed at the diffuser outlet. Major difference between these two models is that in the numerical model the complete non-linear Navier-Stokes equations are solved, while in the two-dimensional inviscid analysis the disturbance is assumed to be linear. According to the two-dimensional inviscid flow analysis, the rotating stall flow field should be a disturbance on top of the assumed steady flow. This is physically possible in the beginning phase of the instability. Therefore, the two-dimensional inviscid flow analysis is only suitable to study the core-flow instability, and is not suitable to study the rotating stall flow field. This analysis is only interesting to determine which modes are the most unstable and when. In the current numerical model, the flow field is determined by the physical instability, which will be shown below. Furthermore, in the two-dimensional inviscid flow analysis no viscosity effects are taken into account, while in the current numerical model the molecular viscosity is used.

### 4.2 Impeller startup

The stable and unstable operating flow conditions, as introduced in section 2.3, are generated by prescribing a step function to the impeller velocity. The impeller tip speed goes from zero to the desired impeller tip speed in one time step, then it remains constant. This situation is illustrated in figure 4.1, where the impeller speed and the corresponding solutions are given as a function of time.

The solution drastically changes when the impeller speed is gradually increased to the desired value. In figure 4.2, first the slow startup and then a fast startup is applied. In this figure, the impeller speed and the corresponding solutions are also given as a function of time. Low impeller acceleration is applied at \( t = 0 \) [s], while high impeller acceleration is applied at \( t = 1 \) [s]. The low acceleration of the impeller towards the desired impeller speed, results in a solution as shown in figure 4.2 at \( t = 0.4 \) [s] and \( t = 0.7 \) [s]. Here, the solution has a velocity field that gradually decreases in strength towards the outlet. By increasing the impeller speed again with high acceleration of the impeller, which is done at \( t = 1.0 \) [s], conditions can be generated, which correspond to the stable or unstable operating flow conditions, as obtained at \( t = 1.17 \) [s], \( t = 1.49 \) [s] and \( t = 1.7 \) [s]. This means that only high acceleration of the impeller leads to the conditions where the two-dimensional rotating instability exists. Low acceleration does not lead to any instability since a nicely distributed velocity field is obtained. Figure 4.2 shows that the instability can also be generated when another initial condition is used than the zero velocity initial condition shown in figure 4.1.

In order to obtain the desired operating flow conditions, the time period in which the acceleration takes place \( \Delta t \) has to be smaller than the critical time constant, \( \Delta t < \Delta t_{cr} \). Hence, \( \Delta t_{cr} \) is determined for different operating flow conditions and diffuser geometries. From extensive calculations, summarized in table 4.1, the critical time constant can be determined according to,

\[
\Delta t_{cr} \approx \frac{r_3}{\Delta V_2}
\]

where \( V_2 \) is the velocity magnitude at the diffuser inlet, \( r_3 \) the diffuser outlet radius and \( \Delta V_2 \) is...
4.2. IMPELLER STARTUP

\[ \Omega R = 2.55 \text{ rev} t = 0.6 \text{ s} - 6.28 \times 10^6 \cdot Re = 0.6 \text{ s} - 1.0 \text{ s} \]

Figure 4.1: Simulation illustrating a fast impeller startup for \( r_3/r_2 = 1.52 \) at \( \alpha_m = 6.8^\circ \) by applying a step function at \( t = 0 \text{ s} \) and fast startup is applied at \( t = 1 \text{ s} \).

Figure 4.2: Simulations illustrating slow and fast impeller startup for \( r_3/r_2 = 1.52 \) at \( \alpha_m = 6.8^\circ \), where slow startup is applied at \( t = 0 \text{ s} \) and fast startup is applied at \( t = 1 \text{ s} \).
### Table 4.1: Numerically determined $\Delta t_{cr}$ and Strouhal number for different geometries and operating flow conditions

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Case 1</th>
<th>Case 2</th>
<th>Case 3</th>
<th>Case 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>$r_3$</td>
<td>0.4908</td>
<td>0.4908</td>
<td>0.04908</td>
<td>0.04908</td>
</tr>
<tr>
<td>$\Delta V_2$</td>
<td>8.668</td>
<td>2.714</td>
<td>8.668</td>
<td>2.714</td>
</tr>
<tr>
<td>$\Delta t_{cr}$ based on equation 4.1</td>
<td>$0.057 \approx 0.181$</td>
<td>$0.0057 \approx 0.0181$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$\Delta t_{cr}$ obtained with numerical model</td>
<td>$0.006 \approx 0.19$</td>
<td>$0.0075 \approx 0.02$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>$f$</td>
<td>4.253</td>
<td>1.331</td>
<td>42.53</td>
<td>13.31</td>
</tr>
<tr>
<td>$St$</td>
<td>$\approx 0.24 \approx 0.24 \approx 0.24 \approx 0.24$</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

the applied step in velocity magnitude. The corresponding Strouhal number can be defined as,

$$St = \frac{f r_3}{\Delta V_2}$$

(4.2)

with $f$ the impeller rotation frequency. Then, for Strouhal numbers $St > 0.24$ the large-scale instabilities are to be expected.

The stable and unstable operating flow conditions seem to be generated only when the radial direction contains a high discontinuity in the tangential velocity component. This discontinuity results into a high-velocity region near the impeller and low-velocity region near the diffuser outlet. In figure 4.3, it is shown how the flow field develops in time, just after the impeller rotation is started. Here, the contours of velocity and vorticity magnitude are given as a function of time. At the separation line between the high-velocity and the low-velocity region, a shear-layer instability will occur, which results in a growing wavelike disturbance. Depending on the magnitude of the inlet flow angle, the wavelike disturbance will transform into the stable operating flow condition or the two-dimensional rotating instability.

### 4.3 Jet-wake interaction

To investigate the influence of the jet-wake shape and intensity on the two-dimensional rotating instability, a number of jet-wake parameters are varied. This is done by varying only one parameter at the time, with respect to the reference diffuser geometry and operating flow conditions given in table 2.1. To judge the influence of each parameter, the critical inlet flow angle $\alpha_{cr}$, the number of rotating cells $m$ and the relative propagation speed $\omega_{rs}/\omega_i$ are monitored and analyzed.

**Jet-to-wake intensity ratio** The jet-to-wake intensity ratio $u_{jet}/u_{wake}$ is the ratio of the radial jet velocity versus the radial wake velocity, as illustrated in figure 4.4. The $u_{jet}/u_{wake}$ ratio is varied from 1 to 11.5, where $u_{jet}/u_{wake} = 1$ represents the uniform velocity profile where no jet-wake pattern is present. The critical inlet flow angle, the number of rotating cells and their propagation speed, obtained for different $u_{jet}/u_{wake}$ ratios are plotted in figure 4.5.

The $u_{jet}/u_{wake}$ ratio has no influence on the critical inlet flow angle until the low $u_{jet}/u_{wake}$ ratios are reached, $u_{jet}/u_{wake} < 3$. At low $u_{jet}/u_{wake}$ ratios the critical inlet flow angle slightly increases. Even when $u_{jet}/u_{wake} = 1$ the two-dimensional rotating instability is observed. This means that the jet-wake pattern at the diffuser inlet is not essential for the occurrence of the two-dimensional rotating instability. Since there is no jet-wake at the diffuser inlet when $u_{jet}/u_{wake} = \ldots$
Figure 4.3: Upper row represents contours of velocity magnitude as a function of time and bottom row represents contours of vorticity, showing the growth of the wavelike disturbance and transition into the rotating cells, $r_3/r_2 = 1.52$, $\alpha_m = 6.8^\circ$.
1, a stable operating flow condition is difficult to define. There is no alternating flow pattern near the diffuser outlet, but the flow is homogeneously distributed around the circumference and the velocity magnitude gradually decreases in the radial direction towards the diffuser outlet. At $u_{jet}/u_{wake} = 1$, the critical inlet flow angle is higher than for other $u_{jet}/u_{wake}$ ratios, $\alpha_{cr} = 14.5^\circ$, but the number of cells and their propagation speed remain the same as for $u_{jet}/u_{wake} < 3$.

According to figure 4.5 the core-flow seems to be less stable at small $u_{jet}/u_{wake}$ ratios, which indicates that the jet-wake has a stabilizing effect on the vanless diffuser core flow. Figure 4.5 shows that the critical inlet flow angle and the number of rotating cells both slightly increase with the decreasing $u_{jet}/u_{wake}$ ratio, but the propagation speed of approximately 40 % of the impeller speed remains unchanged. The number of cells increases from $m = 7$ to $m = 8$ as the jet-wake vanishes, but it still corresponds to the estimated maximum value for this diffuser geometry.

**Wake-to-jet circumferential extent ratio** The wake-to-jet circumferential extent ratio $\theta_{wake}/\theta_{jet}$ is the ratio of the circumferential extent of the wake with respect to that of the jet, as defined in figure 4.6.

The behavior of the two-dimensional rotating instability is studied for the $\theta_{wake}/\theta_{jet}$ ratios of 1, 1.8 and 3.7. To make sure that the mass flow rate remains unchanged when varying the $\theta_{wake}/\theta_{jet}$ ratio, the mean radial velocity $u_m$ is also increased along with $\theta_{wake}/\theta_{jet}$. The influence of the $\theta_{wake}/\theta_{jet}$ ratio on the critical inlet flow angle, number of rotating cells and their propagation speed is shown in figure 4.7. The critical inlet flow angle and the number of rotating cells slightly increase as the $\theta_{wake}/\theta_{jet}$ ratio decreases, while the propagation speed of the cells remains at approximately 39 % of the impeller speed. The influence of the $\theta_{wake}/\theta_{jet}$
4.3. JET-WAKE INTERACTION

Figure 4.5: Critical inlet flow angle, number of rotating cells and their relative propagation speed versus $u_{\text{jet}}/u_{\text{wake}}$ ratio, obtained with the two-dimensional core-flow model for $r_3/r_2 = 1.52$.

Figure 4.6: Illustration of the varied wake-to-jet circumferential extent ratio $\theta_{\text{wake}}/\theta_{\text{jet}}$. 
ratio is noticeable but is not as large as the influence of the diffuser radius ratio or the number of impeller blades.

**Jet-wake steepness** The steepness of the peaks of the jet-wake function, which is represented by the constant $D$ in equation 2.10, is illustrated in figure 4.8. Calculations are performed for three different values of $D$, namely $D = 5$, $D = 20$ and $D = 1000$.

Figure 4.9 shows the critical inlet flow angle, number of rotating cells and their propagation speed as a function of the jet-wake steepness $D$. The change of the critical inlet flow angle with varying jet-wake steepness $D$ is negligible with respect to the influence of previously varied parameters on the critical inlet flow angle. The number of rotating cells remains unchanged and their propagation speed fluctuates around 40% of the impeller speed. According to figure 4.9, stability of the vaneless diffuser core flow is hardly influenced by the jet-wake steepness.

**Velocity fluctuations** Velocity fluctuations are added to the radial velocity component and varied in intensity as illustrated in figure 4.10. The added velocity fluctuations have the intensities of 1%, 5% and 10% of the mean radial velocity component. To study the influence of additional high-wave number fluctuations on the two-dimensional rotating instability, simulations without additional fluctuations are compared with simulations where fluctuations of different intensities are added. Figure 4.11 shows the influence of the additional velocity fluctuations on the critical inlet flow angle, number of rotating cells and their propagation speed. Changes in all three parameters are negligibly small, which indicates that additional velocity fluctuations have no influence on the behavior and characteristics of the two-dimensional rotating
Figure 4.8: Illustration of the varied jet-wake steepness $D$

Figure 4.9: Critical inlet flow angle, number of rotating cells and their relative propagation speed versus transition steepness constant $D$, obtained with the two-dimensional core-flow model for $r_3/r_2 = 1.52$
instability.

4.4 Tangential acceleration

Due to the rotation of the impeller body the fluid particles in the $r - \theta$ plane of the diffuser space experience tangential acceleration due to the curvature of the streamline and acceleration orthogonal to the tangent of the trajectory. To determine if the tangential acceleration might be responsible for the occurrence of the two-dimensional rotating instability, the Taylor number is introduced. This non-dimensional number represents the ratio between the tangential forcing and the viscous drag force in the Taylor problem as described in Kundu [35]. Here, the Taylor number is defined as,

$$ Ta = \frac{\Omega^2 r_m d_0^2}{\nu^2} $$

(4.3)

where $\Omega$ is the angular velocity of the inner rotating cylinder, $r_m$ is the average radius defined as $r_m = (r_3 + r_2)/2$, $d_0$ is the distance between the inner rotating cylinder and the outer cylinder and $\nu$ is the kinematic viscosity of the fluid. Since the instability of the flow between two rotating cylinders appears as a patch of structures in a $z - r$ plane, it is not directly comparable to the two-dimensional rotating instability in the $r - \theta$ plane as obtained with the current numerical model. Therefore, the Taylor number is only used as an indication for the magnitude of tangential acceleration effect.

To estimate the Taylor number for the current numerical model, the outlet boundary condition is considered as the outer cylinder. The Taylor number is estimated for the scaled operating flow conditions and geometry as given in table 2.1, where the jet-wake is prescribed at the inlet.
4.4. TANGENTIAL ACCELERATION

The corresponding Taylor number is found to be $Ta = 2.8 \cdot 10^{10}$. Since the Taylor number in the current numerical model is quite high, it is appropriate to assume that the tangential acceleration might be considered as a possible cause of the two-dimensional rotating instability.

To determine the importance of tangential acceleration on the core flow instability, the earlier found stability criterion is rewritten in terms of the Taylor number. The rotating cells occur when the condition $\alpha_m < \alpha_{cr}$ is satisfied. The mean flow angle can be written in terms of the Taylor and the Reynolds number as follows,

$$\frac{1}{\tan \alpha_m} = \frac{U_2}{u_{m2}} \sim \sqrt{\frac{Ta}{Re^2}} \sim \frac{\Gamma}{Q}, \quad (4.4)$$

where the Reynolds number is defined as,

$$Re = \frac{r_2 u_{m2}}{\nu}, \quad (4.5)$$

and where $Q$ is the volume flow rate per unit of diffuser width and $\Gamma$ is the net circulation, as defined in appendix A. According to equation 4.4, the condition

$$Re < \tan (\alpha_{cr}) \sqrt{Ta}. \quad (4.6)$$

needs to be satisfied in order to obtain instability.

If tangential acceleration causes the two-dimensional rotating instability, the instability should also occur in a situation where no-jet-wake is prescribed, and no base flow with a shear-layer is present. To investigate this, two-simulations are performed where no-jet wake and zero
mass flow rate through the diffuser are prescribed. Only a rotating tangential velocity at the diffuser inlet is prescribed. This is done for two outlet boundary conditions, where one is constant pressure and the other is no-slip wall condition. The solutions obtained with these two situations are given in figure 4.12. According to figure 4.12, in both situations no instability did occur. Since in this case
\[ Re = \sqrt{T_a \tan (\alpha_m)} = 0 < \sqrt{T_a \tan (\alpha_{cr})}, \quad (4.7) \]
the instabilities are to be expected, which leads to a contradicting outcome.

One possible reason for this outcome could be that the instability requirement does not hold in the limit \( \alpha \to 0 \). According to the model of Tsujimoto et al. [59], this is quite likely. Here, the condition \( \phi = 0 \) can not be modeled unless \( \omega_i \) also equals zero. In case of a physical solution, the convective forces in the radial direction are probably so small that the flow needs much more time to develop. Therefore, this flow remains stable within the short time span in which the two-dimensional rotating instability occurs. Since in this case, nor the jet-wake nor the through-flow is prescribed at the diffuser inlet, there is no base flow entering the diffuser space. Hence, there is no large disturbance to trigger the onset of instability.

### 4.5 Shear-layer between in- and outlet flow region

To study the influence of the outlet boundary condition, the constant-pressure boundary condition is replaced by the no-slip wall boundary condition at the diffuser outlet, which approaches the situation in section 3.3. When no-slip wall boundary condition is used at the outlet, no through-flow is possible, which means that the mass flow rate in this case must equal zero. The jet-wake pattern is still applied by prescribing the amplitude of a nonzero value to the hyperbolic tangent function. In figure 4.13, the upper solutions are obtained with the no-slip wall boundary condition, and the lower solutions are obtained with the constant-pressure boundary condition. The solutions are given for \( t = 0.1 \, [s] \), \( t = 0.2 \, [s] \), \( t = 0.3 \, [s] \) and \( t = 0.4 \, [s] \). For better comparison, the solutions obtained with constant-pressure boundary condition are obtained for zero through-flow, which explains why the cells propagate very closely to the impeller. Increase of the mean radial velocity component results in a somewhat larger distance of the cells from the impeller.

The solutions given in figure 4.13 show that the two-dimensional rotating instability will occur when jet-wake and zero through-flow are prescribed. Since the instability occurs at small inlet flow angles, the condition \( \alpha_m < \alpha_{cr} \) is always satisfied in this case. Major difference between the two situations in figure 4.13 is found to be the life-time of the cells. The rotating cells obtained with the no-slip wall boundary condition at the outlet disappear after some time, while in the other case a few cells continue to exist and propagate around the circumference. This dissipation effect is due to the presence of the solid wall, which inhibits the fluid motion to a certain degree. But it is also shown that, on the short time scale at which the two-dimensional rotating instability is studied, in both cases the two-dimensional rotating instability will occur. Therefore, it is appropriate to assume that as long as a shear-layer is present between the jet-wake pattern and the low velocity fluid near the outlet, the two-dimensional rotating instability will occur.

### 4.6 Effect of viscosity

In this section, the influence of viscosity is illustrated, by application of the engineering turbulence models, by increasing the molecular velocity magnitude and by varying the Reynolds
4.6. EFFECT OF VISCOSITY

**Figure 4.12:** Simulations for $r_3/r_2 = 1.52$ with no-through-flow, $\alpha_m = 0$, and no jet-wake $u_{jet}/u_{wake} = 1$, obtained for two outlet boundary conditions, on the left constant pressure and on the right no-slip wall condition.

**Figure 4.13:** Comparison between different outlet boundary conditions for $r_3/r_2 = 1.52$ and $\alpha_m = 0$; upper row represents the solutions with solid wall at the outlet and bottom row represents solutions with constant pressure at the outlet.
number at the diffuser inlet.

**Turbulence modeling**  To determine the influence of turbulent viscosity on the two-dimensional rotating instability, results obtained with the standard $k - \varepsilon$ and RNG $k - \varepsilon$ model available in Fluent are compared with the results obtained with the incompressible laminar flow model.

As opposed to the laminar models or DNS, engineering turbulence models account for the turbulent dissipation effect, because turbulent or eddy viscosity is used instead of only the molecular viscosity. This dissipation effect is slightly overestimated because turbulence models tend to add excessive numerical dissipation to the solutions, see Wesseling [61].

To test the influence of engineering turbulence models, the operating flow conditions are set such that the two-dimensional rotating instability is obtained. Solutions obtained with the engineering turbulence models and applied initial flow conditions are given in figure 4.14, where solutions are represented by contours of velocity magnitude. For each turbulence model two simulations were performed with different initial conditions. The two solutions on the left in figure 4.14 represent the two initial conditions, where the upper one is the zero velocity initial condition and the bottom initial condition is the solution obtained with the viscous laminar flow model. All the solutions shown in figure 4.14, except the initial conditions, are obtained at $t = 0.6$ [s] after the computation is started. Figure 4.14 shows that for both turbulence models a similar result is obtained. The two-dimensional rotating instability does not occur, when zero velocity field was used as initial guess, and the two-dimensional rotating cells are totally dissipated, when the solution obtained by the laminar viscous flow model was used as initial guess.

The turbulent eddy viscosity in the turbulence models is compared to the molecular viscosity of the incompressible viscous flow model. The mean turbulent viscosity in the standard $k - \varepsilon$ and RNG $k - \varepsilon$ model was $\mu_t = 100.3$ [Pa s] and $\mu_t = 21.2$ [Pa s] respectively when two-dimensional rotating instability was used as initial guess, and $\mu_t = 49.5$ [Pa s] and $\mu_t = 12.6$ [Pa s] when zero velocity was used as initial guess. Compared to the molecular viscosity of water $\mu = 0.001$ [Pa s], the obtained turbulent viscosity with the turbulence models is $10^4 - 10^5$ times higher.

**Molecular viscosity**  To illustrate the influence of additional viscosity on the two-dimensional rotating instability, simulations are performed with the incompressible viscous flow model with an increased molecular viscosity. Solutions obtained for molecular viscosities of $\mu = 0.01$ [Pa s], $\mu = 0.1$ [Pa s], $\mu = 0.2$ [Pa s], $\mu = 0.3$ [Pa s], $\mu = 1$ [Pa s] and $\mu = 10$ [Pa s] are given in figure 4.15. Each solution in figure 5.7 is obtained by applying the unstable operating flow conditions to the zero velocity initial guess. This figure shows that a higher amount of viscosity makes it harder for the two-dimensional rotating instability to occur. When the additional viscosity is relatively low, $\mu = 0.01$ [Pa s], $\mu = 0.1$ [Pa s] and $\mu = 0.2$ [Pa s], the two-dimensional rotating instability will occur, but the number of rotating cells decreases. When additional viscosity is much higher, $\mu = 0.3$ [Pa s] and $\mu = 1$ [Pa s], the two-dimensional rotating instability is not able to break down the stable core-flow pattern. And for even larger additional viscosity, $\mu = 10$ [Pa s], all flow structures within the vaneless diffuser space, except the prescribed jet-wake at the inlet, are being fully dissipated.

In addition, the influence of the inlet Reynolds number, defined as $Re = V_2 r_2 / \nu$, is also varied for $\mu = 0.001$. In figure 4.16, the obtained critical inlet flow angles and number of rotating cells are plotted versus the inlet Reynolds number.

The critical inlet flow angle decreases with decreasing Reynolds number, but compared to the influence of the diffuser radius ratio this influence is quite small. Since the viscosity increases
4.6. EFFECT OF VISCOSITY

\[ \dot{\gamma} = 6.8, \quad \theta = 0.6, \quad R = 2.52 \text{ at } \alpha = 1.52, \quad \mu = 0.01, 0.1, 0.2, 0.3, 1.0, 10 \text{ Pa s} \]

Figure 4.14: Influence of the turbulence models for \( r_3/r_2 = 1.52 \) at \( \alpha_m = 6.8^\circ \): a) initial conditions, b) standard \( k-\varepsilon \) model, c) RNG \( k-\varepsilon \) model. The solutions in the upper half are obtained with zero-velocity initial guess and, the solutions in the lower part are obtained with the laminar two-dimensional rotating initial guess.

Figure 4.15: Solutions obtained with the incompressible viscous flow model for \( r_3/r_2 = 1.52 \) at \( \alpha_m = 6.8^\circ, t = 0.6 \text{ s}, R = 2.55 \text{ rev} \), where colors represent contours of velocity magnitude.

\[ \dot{\gamma} = 6.8, \quad \theta = 0.6, \quad R = 2.52 \text{ at } \alpha = 1.52, \quad \mu = 0.01, 0.1, 0.2, 0.3, 1.0, 10 \text{ Pa s} \]
with decreasing inlet Reynolds number, a probably somewhat higher viscosity value contributes to the stability of the vaneless diffuser core flow, which results in lower critical inlet flow angles.

Also the influence of the inlet Reynolds number on the number of rotating cells is studied. Figure 4.16 shows that the number of rotating cells slightly decreases with decreasing Reynolds number. Also here, the increasing viscosity with decreasing Reynolds number, which increases the rate of dissipation, probably causes the number of rotating cells to decrease. Therefore, it is appropriate to assume that a further decrease of the inlet Reynolds number will lead to further stabilization of the flow field and decrease of the number of rotating cells, and that eventually the dissipation rate will become so high to sweep out large flow field structures.

**Recovery**  As shown in figure 2.6, transition from a stable operating flow condition into the two-dimensional rotating instability can be generated by gradually decreasing the mean radial velocity component, which corresponds to the decrease of the mass flow rate. To check if the reversed process is possible, mass flow rate is gradually increased starting from the unstable operating flow condition. This numerical simulation has revealed that the recovery from the unstable operating flow condition into the stable operating flow condition is not possible. This solution does not correspond to practice, where despite the hysteresis eventually a stable operating flow condition is reached. The numerical model can probably not reorganize the flow because of the missing dissipation effects caused by turbulence and wall boundary layers, which contribute to stabilization of the two-dimensional core flow.
4.7 Discussion

The two-dimensional inviscid stability analysis performed by Tsujimoto et al. [59] shows that the two-dimensional rotating flow instability is inviscid. This means that the Reynolds number similarity, discussed earlier during scaling in section 2.1, does not necessarily have to be satisfied.

By varying the impeller startup condition, it is shown that the instability occurs only when it is triggered by the high discontinuity in the tangential velocity component. If the impeller is slowly accelerated, no circumferential disturbances and thus no instability will occur. This means that the instability is primarily generated by the shear-layer between the impeller and diffuser outlet.

The two-dimensional inviscid stability analysis as well as the current numerical model show that the two-dimensional core-flow instability is not a jet-wake instability. Tsujimoto et al. [59] found that the flow instability associated with vaneless diffuser rotating stall may occur even with uniform outward flow. With the current numerical model it is shown that the jet-wake shape and intensity have negligible influence on the two-dimensional rotating instability.

By varying the outlet boundary condition it is shown that the wall-boundary layer influences the life-time and the radial distance of the cells to the impeller. It is also shown that the instability will not occur within a short time scale, when uniform zero-through-flow is prescribed at the inlet. This possibly indicates that the tangential acceleration by itself, is not responsible for the occurrence of the two-dimensional rotating instability. Anyway, it is shown that the tangential forcing plays a dominant role in the origin of the core-flow instability. The ratio between the tangential forcing and convective forces has to be larger than a certain value, which is in this case $1/\tan \alpha_{cr}$, in order to prevent instability. The instability is expected when $Re < \tan (\alpha_{cr}) \sqrt{T/u}$. Since it is shown that the jet-wake pattern is not required to generate the two-dimensional rotating instability, it is appropriate to assume that primarily the shear-layer between the impeller region and the diffuser outlet, together with the tangential acceleration are responsible for the core-flow instability.

Numerical simulations show that the turbulence models available in Fluent fully damp out the large-eddy structures. This effect is most likely caused by the excessive numerical dissipation. Usually, performance of the turbulence models is related to the flow situation that is being modeled. Therefore, they are found to be not suitable for the study of the two-dimensional core-flow instability. On the other hand, the molecular viscosity underestimates the effect of turbulent viscosity. Therefore, it is assumed that in practice a compromise between these two cases is met.

By increasing the magnitude of viscosity in the numerical model, it is shown that addition of the turbulent viscosity results in a lower number of rotating cells and that it has a stabilizing effect on the two-dimensional core flow. Since in the current two-dimensional numerical model the viscosity effect is underestimated, it is appropriate to assume that in practice the number of rotating cells and the critical flow angle would be slightly lower than obtained with the current numerical model. It is assumed that in practice the dissipation rate of the turbulent viscosity is small enough to leave the stronger fully-developed rotating stall cells to exist.
Chapter 5

Influence of geometry

In this chapter the influence of the geometry parameters is investigated. To study the influence of the diffuser geometry, the diffuser radius ratio, the diffuser width ratio and the number of impeller blades are varied. Where possible, the numerical model results are also compared with the experimental data found in the literature. The influence of geometry is analyzed by varying only one parameter at the time, with respect to the reference diffuser geometry given in table 2.1. To judge the influence of each parameter, the critical inlet flow angle $\alpha_{cr}$, the number of rotating cells $m$ and the relative propagation speed $\omega_{rs}/\omega_i$ are monitored and analyzed.

5.1 Diffuser radius ratio

To investigate the influence of the diffuser radius ratio $r_3/r_2$ on the stability of the vaneless diffuser core flow, the diffuser outlet radius is varied while the inlet radius remained unchanged. Diffuser radius ratios $r_3/r_2 = 1.2$, $r_3/r_2 = 1.52$ and $r_3/r_2 = 2.0$ are investigated, see also Ljevar et al. [38, 41]. The influence of the diffuser radius ratio on the critical inlet flow angle $\alpha_{cr}$, number of rotating cells $m$ and their relative propagation speed $\omega_{rs}/\omega_i$ is shown in figure 5.1, where the value of these parameters is plotted versus the diffuser radius ratio.

The critical inlet flow angle is found to be strongly dependent on the diffuser radius ratio. It decreases with decreasing diffuser radius ratio, which contributes to the stability of the vaneless diffuser core flow. According to figure 5.1 the number of rotating cells decreases with increasing diffuser radius ratio. Since the number of rotating cells also decreases with decreasing mass flow rate and corrrsponding $\alpha_m$, different modes were found at each diffuser radius ratio when the mass flow rate was decreased. The mass flow rate decreases in the direction of the arrows. For constant diffuser radius ratio, the propagation speed of the cells does not change with changing number of rotating cells. The propagation speed of the rotating cells is found to decrease when the diffuser radius ratio is increased. The increase in diffuser radius ratio results in a larger distance of the cells from the impeller and thus lower propagation speed of the cells. This is in good agreement with the measurements performed in the wide vaneless diffuser behind the radial flow pump impeller, which is discussed in chapter 3. These measurements have shown that the propagation speed of the rotating stall cells decreases as the distance of the cells from the impeller increases.

In figure 5.2 solutions corresponding to three different diffuser radius ratios are shown, where the upper figures represent the stable operating flow condition and the lower figures represent the two-dimensional rotating instability. This shows that at stable operating flow condition, the alternating pattern near the diffuser outlet becomes smaller when the diffuser radius ratio
Figure 5.1: The numerically found critical inlet flow angle, number of rotating cells and their relative propagation speed plotted versus the diffuser radius ratio. In the middle graph also the outcomes are given for a decreasing value of $\alpha_m$. 
increases. Since the alternating pattern near the diffuser outlet is a result of interaction between the jet-wake pattern prescribed at the inlet and constant pressure condition prescribed at the outlet, it becomes smaller due to the larger distance between the inlet and the outlet boundary condition. Knowing the influence of the diffuser radius ratio on the stable operating flow condition, the influence of the diffuser radius ratio on the critical inlet flow angle, as shown in figure 5.1, can be explained. The higher the diffuser radius ratio, the further away the alternating pattern is from the jet-wake flow at the inlet. Therefore, at larger radius ratios, the inlet flow angle at which the jet-wake flow can pass underneath the alternating pattern and cause instability, is larger as explained in section 2.3. For even larger diffuser radius ratios than shown in figure 5.2, \( \frac{r_3}{r_2} > 2.5 \), the diffuser flow is found to be unstable at all inlet flow angles. Here, the alternating flow pattern near the diffuser outlet is far and small enough, so that the jet-wake flow and the alternating pattern hardly can interact with each other. This always results in the unstable operating flow condition.

Figure 5.2 reveals that not only the number of rotating cells but also their size changes as the diffuser radius ratio is varied. The number of rotating cells decreases as the radius ratio increases, while the size of the cells increases. This influence of the diffuser radius ratio can be explained as follows. As the diffuser radius ratio increases, the radial distance between the diffuser inlet and diffuser outlet becomes larger, which allows the cells to have a larger radial extent. The radial diffuser length is most likely the determinant parameter for the size of the rotating cells. It seems that the circumferential and radial extent of the rotating cells tend to be proportional, which indicates that the maximum number of the cells is probably defined by the circumference of the diffuser. It depends on how many cells of a given size fit into the circumference of the diffuser. If this is the case the maximum number of the cells can be estimated by the ratio between the circumference and the diffuser length,

\[
m = \frac{1}{2} \pi \left( \frac{r_3 + r_2}{r_3 - r_2} \right). \tag{5.1}
\]

The ratio between the circumference and the radial diffuser length is divided by 2 to take into account the space between the two rotating cells. For each rotating cell, an additional space of the same size is taken into account. Using equation 5.1 the maximum number of rotating cells for the diffuser radius ratios \( \frac{r_3}{r_2} = 1.2 \), \( \frac{r_3}{r_2} = 1.52 \) and \( \frac{r_3}{r_2} = 2 \) is estimated to be \( m = 17 - 18 \), \( m = 7 - 8 \) and \( m = 4 - 5 \), respectively. Numerical simulations show that the maximum number of cells occurring near the stability limit for \( \frac{r_3}{r_2} = 1.52 \) varies between 7 and 8 cells, for \( \frac{r_3}{r_2} = 2 \) between 4 and 5 cells and for \( \frac{r_3}{r_2} = 1.2 \) is found to be \( m = 13 \). With this observation it seems that the maximum number of rotating cells is well predicted for \( \frac{r_3}{r_2} = 1.52 \) and \( \frac{r_3}{r_2} = 2 \). The number of rotating cells for \( \frac{r_3}{r_2} = 1.2 \) approaches the predicted value, but is not the same. This is probably due to the fact that the outlet boundary condition and the inlet boundary are too close, which leads to a somewhat suppressed solution that makes this comparison difficult.

The number of rotating cells that is compared with equation 5.1, is always the maximum number of rotating cells, which is obtained with the numerical model shortly after the unstable operating flow condition is reached. Numerical simulations show that after the maximum number of cells has occurred, the number of rotating cells starts to decrease with a further decrease of the mass flow rate. Therefore, the radial diffuser length is not the only parameter influencing the number of rotating cells. It is obvious to assume that the number of rotating cells, their size, their distance from the impeller and impeller speed influence the propagation speed of the rotating instability, but the influence of other parameters was studied first before concluding that these are the only parameters influencing the propagation speed of the cells.
Figure 5.2: Left solutions of the stable operating flow condition, and right solutions of the two-dimensional rotating instability for different diffuser radius ratios.
The numerical model results are compared with the two-dimensional inviscid stability analysis performed by Tsujimoto et al. [59], because they have obtained the relation between the instability characteristics, such as the critical inlet flow angle and the relative propagation speed, and the diffuser radius ratio. Then, the numerical model results are compared with the experimental data found in the literature, where the influence of the diffuser radius ratio is investigated.

5.1.1 Comparison with the two-dimensional inviscid flow model

In this section, the influence of the diffuser radius ratio obtained with the two-dimensional numerical model is compared with the two-dimensional inviscid stability analysis developed by Tsujimoto et al. [59], see also Ljevar et al. [40]. The two-dimensional inviscid stability analysis is already explained in chapter 4.

Tsujimoto et al. [59] have performed the two-dimensional inviscid flow analysis only for the low-order modes, namely \( m = 1 \), \( m = 2 \) and \( m = 3 \). Since in the current numerical model also higher numbers of rotating cells were observed, higher-order mode solutions were desired for comparison. To extend the range with the higher-order modes, calculations were repeated, and solutions for the higher-order modes \( m = 4 \) to \( 18 \) were obtained. The solutions for \( m = 1 \) to \( 18 \) are shown in figures 5.3 and 5.4. In figure 5.3 the critical inlet flow angle, and in figure 5.4 the relative propagation speed of the disturbance is plotted versus the diffuser radius ratio.

For arbitrarily chosen diffuser radius ratio and mode number in figure 5.3, the flow is stable when the flow angle is higher than the critical flow angle and unstable when the flow angle is lower than the critical flow angle. Therefore, the mode number that has the highest critical inlet flow angle is considered as the most unstable mode for a given diffuser radius ratio. It follows from figure 5.3 that the most unstable modes for \( r_3/r_2 = 1.2 \), \( r_3/r_2 = 1.52 \) and \( r_3/r_2 = 2 \) are \( m = 9 \), \( m = 5 \) and \( m = 3 \) respectively.

The results obtained with the two-dimensional numerical model are compared with the results obtained with the two-dimensional inviscid flow analysis. Therefore, the maximum-order modes or the most-unstable modes obtained by the two-dimensional inviscid flow analysis are compared with the first occurring mode number after instability inception in the two-dimensional numerical model. In the two-dimensional numerical model, instability occurrence due to the mass flow decrease from the stable operating flow condition is considered. Since the maximum occurring mode numbers obtained with the current numerical model are, \( m = 13 \), \( m = 6 - 7 \) and \( m = 4 - 5 \), and obtained with the two-dimensional inviscid analysis are \( m = 9 \), \( m = 5 \) and \( m = 3 \), for \( r_3/r_2 = 1.2 \), 1.52 and 2 respectively, a quite good agreement is found between the two models. This comparison is shown in figure 5.5, where the most unstable modes obtained with both models are plotted versus the diffuser radius ratio. In both cases, a growing trend of the mode number is obtained as the diffuser radius ratio decreases.

The critical inlet flow angles from figure 5.3 and the propagation speeds from figure 5.4, corresponding to the maximum mode number for a given diffuser radius ratio, are also compared with the critical inlet flow angles and propagation speeds obtained with the two-dimensional core-flow model. Comparison of the critical inlet flow angles and the propagation speed of the cells is given in figure 5.6, where their values are plotted versus diffuser radius ratio. Figure 5.6 shows that very good agreement between the two models is obtained.

5.1.2 Comparison with experimental data from literature

According to the numerical model results shown in section 5.1, the diffuser radius ratio has a significant influence on the critical inlet flow angle, the number of rotating cells and the propagation speed of the cells. To check the validity of the two-dimensional core-flow model once
Figure 5.3: Critical inlet flow angle versus diffuser radius ratio for $m = 1 - 18$, obtained with the two-dimensional inviscid analysis

Figure 5.4: Relative propagation speed versus diffuser radius ratio for $m = 1 - 18$, obtained with the two-dimensional inviscid analysis
5.1. DIFFUSER RADIUS RATIO

Figure 5.5: Influence of diffuser radius ratio on the number of rotating cells, comparison between the current numerical model and the two-dimensional inviscid flow analysis

Figure 5.6: Influence of diffuser radius ratio on the critical inlet flow angle and propagation speed, comparison between the numerical model and the two-dimensional inviscid analysis
more, the obtained results are also compared with the measurements found in the literature. Because Abdelhamid [2], Tsurusaki et al. [60], Abdelhamid and Bertrand [3], Abidogun and Ahmed [7] and Abidogun [5, 6] have used relatively wide diffusers, which are not pinched at the diffuser inlet, and because they have measured the rotating stall characteristics for different diffuser radius ratios, their data were used for comparison with the numerical model results.

In figure 5.7, the obtained critical inlet flow angles for different diffuser radius ratios are compared with the measurements performed by Abdelhamid [2], Tsurusaki et al. [60], Abidogun and Ahmed [7] and Abidogun [6], and with the critical inlet flow angles obtained with the two-dimensional inviscid flow analysis. In the legend of figure 5.7 the diffuser width ratio $h/r_2$, corresponding to each reference with measurement data, is given.

Besides good agreement between the two models, figure 5.7 also shows a good agreement with the measurements found in the literature. In all cases a growing trend of the critical flow angle is observed when the diffuser radius ratio increases, and the critical inlet flow angles are of the same order. Measurements of Abdelhamid [2] showed successive occurrence of two rotating pressure patterns, which are in figure 5.7 indicated as $HS$ when high speed rotating patterns and $LS$ when low speed rotating patterns are observed. In addition to figure 5.7, measurements performed by Abdelhamid and Bertrand [3] and Abidogun [6] have also shown that the critical flow coefficient, which is proportional to the critical inlet flow angle, is larger for larger diffuser radius ratios.

According to the current two-dimensional numerical model, the number of rotating cells decreases with increasing diffuser radius ratio. As shown in figure 5.8, the measurements of Abdelhamid and Bertrand [3] show the same trend as that of the numerical model. After varying the diffuser radius ratio Abidogun [6] has found one or two rotating stall cells for all diffuser

Figure 5.7: Influence of the diffuser radius ratio on the critical inlet flow angle, comparison with the data from the literature
5.1. DIFFUSER RADIUS RATIO

radius ratios. Figure 5.8 shows that the number of cells predicted by the current numerical model is much larger than the measured number of rotating stall cells. This difference can be ascribed to several effects such as the wall-boundary layer effects, different impeller inflow pattern and/or the turbulent viscosity effects, because these effects are not taken into account in the two-dimensional core-flow model.

The relative propagation speed of the rotating cells is plotted versus diffuser radius ratio and compared with the data from literature in figure 5.9. For one diffuser radius ratio, the propagation speed of the cells remains unchanged as the number of rotating cells changes. Measurements of Abdelhamid [2], Abdelhamid and Bertrand [3], Abidogun [5] and Abidogun [6] indicate that the propagation speed of rotating stall cells depends on the diffuser radius ratio. They all have found that the propagation speed of the rotating stall pattern decreases with increasing diffuser radius ratio, which is in good agreement with the two-dimensional numerical model and the two-dimensional inviscid flow analysis.

5.1.3 Pressure fluctuations analysis

The dynamics of unsteady flow phenomena such as rotating stall are usually investigated by studying the pressure signatures, measured with pressure transducers or microphones. In this section and in Ljevar et al. [42], the pressure traces obtained by the two-dimensional numerical model are compared with the experimental data. Also the influence of the diffuser radius ratio on the pressure fluctuations is investigated.

To check if the dynamical behavior of the two-dimensional rotating instability obtained with the numerical model is similar to the experimentally observed flow dynamics, pressure traces are compared. Therefore, the static pressure signal is monitored during transition from stable
Figure 5.9: Influence of the diffuser radius ratio on the relative propagation speed, comparison with the data from the literature

into unstable operating flow condition, as shown in figure 5.10, which is achieved by gradually decreasing the mass flow rate through the diffuser. In the upper plot in figure 5.10, the course of the flow coefficient $\phi$ and the mean inlet flow angle $\alpha_m$ is plotted versus time. In the lower plot in figure 5.10, the monitored pressure signal in the middle of the diffuser space at $r = (r_2 + r_3)/2$, is plotted versus time. When the critical inlet flow angle $\alpha_{cr}$ of approximately $10.5^\circ$ is reached, the operating flow condition starts to become unstable, and eventually the two-dimensional rotating instability occurs.

For comparison, in figures 5.11 and 5.12 experimentally obtained pressure traces are shown, which are measured with the flush-mounted pressure transducers. Pressure traces measured by Abdelhamid [4] in the vaneless diffuser space during rotating stall inception are given in figure 5.11. Pressure traces measured in the vaneless diffuser space behind the radial-flow-pump impeller are given in figure 5.12. Static pressure signals shown in figure 5.12 are measured at six different flow rates and the pressure sensors were positioned at $r_{p1} = 0.297$ [m], as discussed in chapter 3. At all flow rates, except for the highest flow rate, rotating stall is observed in the vaneless diffuser space.

Comparison between pressure traces from figures 5.10 to 5.12 indicates that in all three cases the amplitude during rotating stall instability is larger than the amplitude during stable operating flow condition. Since the pressure amplitudes are influenced by the impeller outflow, diffuser radius ratio, diffuser width and outlet boundary condition, which are different for all the three cases, they can not be quantitatively compared. Furthermore, in the experimental cases the pressure traces are measured flush at the wall, while in the numerical case the fluctuating pressure signal at mid-height of the diffuser space is represented, which could lead to differences in fluctuation amplitudes. Therefore, only qualitative comparison of the pressure traces in
Figure 5.10: Data obtained with the two-dimensional numerical model during the instability inception for $r_3/r_2 = 1.52$; the mass flow rate is decreased, while the impeller speed corresponding to $Re = 2.78 \cdot 10^6$ remains unchanged; on the top flow parameters, and at the bottom monitored static pressure signal versus time; $\alpha_{cr} \approx 10.5^\circ$
Figure 5.11: Measured static pressure in the vaneless diffuser space during rotating stall inception where \( r_3/r_2 = 1.51 \) and \( \alpha_{cr} \approx 13^\circ \), after Abdelhamid [4]

figures 5.10 to 5.12 is made. Figures 5.10 to 5.12 shows that the numerically obtained static pressure signals during transition from stable to unstable operating flow-condition look alike the measured static pressure fluctuations in the real vaneless diffuser during the rotating stall inception.

To study the influence of the diffuser radius ratio on the pressure fluctuation, the monitored pressure signals for different diffuser radius ratios are analyzed and discussed. Pressure signals are monitored during the transition from stable into unstable operating flow conditions. They are monitored in the middle of the diffuser, which corresponds to the radial distance \( r = (r_3 + r_2)/2 \). The core-flow instability is generated by gradually decreasing the mean radial velocity component, which corresponds to a decrease of the mass flow rate through the diffuser.

Monitored pressure signals for \( r_3/r_2 = 2 \), \( r_3/r_2 = 1.52 \) and \( r_3/r_2 = 1.2 \) are shown in figures 5.13, 5.14 and 5.15 respectively.

The pressure traces in figures 5.13, 5.14 and 5.15 clearly show the gradual transition from the stable operating flow condition into the two-dimensional rotating instability. In figure 5.13, stable operating flow condition ranges from \( t = 0.2 \) [s] to \( t = 1 \) [s], and then the two-dimensional rotating instability starts to develop. The instability is fully developed at approximately \( t = 1.5 \) [s]. In figure 5.14, the two-dimensional rotating instability starts to develop at \( t = 0.6 \) [s] and is fully developed at approximately \( t = 0.8 \) [s]. Finally, for \( r_3/r_2 = 1.2 \) the pressure signature is given in figure 5.15, where the transition from stable operating flow condition into the two-dimensional rotating instability occurs at approximately \( t = 2.17 \) [s].

After the instability has occurred, mass flow rate continues to decrease as a function of time. Therefore, the amplitude of the pressure fluctuations and the average pressure continue to slightly change with time, as shown in figures 5.13 to 5.15. For \( r_3/r_2 = 2 \) the amplitude of the pressure fluctuations slightly increases as the mass flow rate goes to zero. For \( r_3/r_2 = 1.52 \) the fluctuations amplitude almost remains unchanged, while for \( r_3/r_2 = 1.2 \) it decreases with decreasing mass flow rate. This is probably due to the shift of the rotating cells towards the diffuser inlet. As the mass flow rate decreases, rotating cells move closer to the diffuser inlet.
Figure 5.12: Measured static pressure in the vaneless diffuser space behind the radial-flow-pump impeller at different flow rates, $r_3/r_2 = 1.52$, $h/r_2 = 0.15$
Figure 5.13: Pressure signal during instability inception for $r_3/r_2 = 2$; numerical model; $\alpha_{cr} \approx 13^\circ$.

Figure 5.14: Pressure signal during instability inception for $r_3/r_2 = 1.52$; numerical model; $\alpha_{cr} \approx 10.5^\circ$. 

\[ p_{st}/(\rho U^2) \]
Because the radial velocity component is smaller and thus less powerful to blow the cells away from the impeller. Depending on the position of the cells with respect to the pressure measurement point, the pressure fluctuation will decrease or increase with decreasing mass flow rate. Since the rotating cells have a nearly circular shape, the largest pressure fluctuations are monitored when the center of the rotating cells passes the pressure measurement point. If only the tip of the rotating cell passes the pressure measurement point, the amplitude of the pressure fluctuations is lower. If the center of the rotating cells is closer to the diffuser outlet than the diffuser inlet, the amplitude of the pressure fluctuations increases as rotating cells move towards the impeller. This is the case when \( r_3/r_2 = 2 \), as shown in figure 5.13. If the center of the rotating cells is closer to the diffuser inlet than the diffuser outlet, the amplitude of the pressure fluctuations decreases with further decreasing mass flow rate. This is the case when \( r_3/r_2 = 2 \), as shown in figure 5.15. In figure 5.14, the center of the rotating cells moves closely to the position where the pressure is monitored. Therefore, no clear change in the amplitude of the pressure signal is noticed between \( t = 0.8 \) [s] and \( t = 2 \) [s]. For \( t \geq 2 \) [s], a slight decrease of the pressure amplitude in time is noticeable. This indicates that, on its way towards the diffuser inlet, the center of the rotating cells has just passed the pressure measurement point. The decreasing amplitude of the pressure fluctuations when the rotating cells approach the diffuser inlet is probably also caused by a decrease of the size of the cells as the mass flow rate is being decreased.

In figure 5.16, the relative amplitude \( A_{rs}/A_{st} \), obtained from figures 5.13 to 5.15, is plotted versus diffuser radius ratio. Here, \( A_{rs} \) represents the amplitude of the fluctuations during instability, and \( A_{st} \) represents the amplitude of the fluctuations during stable operating flow.
condition. Figure 5.16 shows that as the diffuser radius ratio decreases, the amplitude of the fluctuations at stable operating flow condition becomes larger relative to the amplitude during instability. The relation between the relative amplitude and the diffuser radius ratio, as shown in figure 5.16, is probably due to the available diffuser space. For larger diffuser radius ratios the alternating pattern near the diffuser outlet is relatively small and further away from the diffuser inlet than for small diffuser radius ratios, as shown in figure 5.2. For \( r_3/r_2 = 1.2 \), the diffuser inlet and diffuser outlet are very close to each other, and the alternating pattern as well as the rotating cells during instability are both located nearly in the middle of the vaneless diffuser space.

The relative amplitude of the measurements performed in the wide vaneless diffuser behind the radial flow pump impeller, shown in figure 5.12, is estimated at \( A_{rs}/A_{st} \approx 3 \). Compared to the numerical model results shown in figure 5.16, this relative amplitude compares better to the numerical model with the diffuser radius ratio \( r_3/r_2 \approx 1.8 \) than with \( r_3/r_2 = 1.52 \), which is the designed value. This is because the outlet boundary condition in this experimental setup is not affecting the flow exactly at the diffuser outlet, but slightly outside of the vaneless diffuser space. Therefore, the vaneless diffuser in this experimental setup can be considered as a diffuser with somewhat larger diffuser radius ratio than initially designed.

### 5.2 Diffuser width ratio

In practice, vaneless radial diffusers have a finite width. This means that wall-boundary layers near the diffuser walls exist, which may influence the two-dimensional core flow. In this section, the effect of wall-boundary layers is investigated along with the influence of the diffuser width on the two-dimensional core-flow instability. The influence of the wall-boundary layers is first modeled using the quasi-steady approach, which is later corrected for the unsteady performance of the wall-boundary shear layer. Finally, numerical observations are compared with experimental evidence found in the literature about the influence of the diffuser width on the core-flow instability.
5.2. DIFFUSER WIDTH RATIO

5.2.1 Quasi-steady viscous contribution

The quasi-steady viscous contribution of the wall boundary layers is implemented into the two-dimensional numerical model by adding an extra source term to the momentum equations. This additional source term represents the overestimated value of the friction force per unit volume, because it is estimated for the case of a fully developed Poiseuille flow profile. For laminar flows, the friction force per unit volume $S$ is equal to

$$S = \frac{\partial \tau_w}{\partial z} = \mu \frac{\partial^2 u}{\partial z^2} \sim \frac{U_\infty}{h^2}, \quad (5.2)$$

where $\tau_w$ is wall shear stress, $z$ diffuser width direction, $\mu$ dynamic viscosity, $U_\infty$ velocity of the flow outside the wall boundary layers and $h$ diffuser width.

To study the influence of the wall boundary layers, the current numerical model is made quasi-three-dimensional. This is obtained by adding the extra source term defined in equation 5.2 to the momentum equations. Therefore, simulations with the reference diffuser geometry and operating flow conditions are performed, where the diffuser width $h$ is varied. The reference diffuser geometry has a diffuser radius ratio $r_3/r_2 = 1.52$ where the corresponding diffuser width ratio is $h/r_2 = 0.025$ and the impeller tip speed in terms of the Reynolds number is $Re = 2.78 \cdot 10^6$. In addition, the diffuser width is also varied for a few other impeller tip velocities than the reference velocity, to study the influence of the Reynolds number based on the diffuser width,

$$Re_h = V h/\nu, \quad (5.3)$$

where $V$ is velocity magnitude and $\nu$ kinematic viscosity. The results are given in figure 5.17, where solutions are represented by the velocity magnitude. The diffuser width is varied in the vertical direction, while columns represent different impeller tip velocities prescribed at the diffuser inlet. The first row contains the solutions calculated with the two-dimensional numerical model as described in the previous chapter, which are given for comparison.

When comparing the solutions for $h/r_2 = 0.025$ with the solutions for $h/r_2 = \infty$, no difference is found, but figure 5.17 shows that for smaller diffuser widths the solutions starts to change. When the diffuser width becomes small enough, the two-dimensional vortex structures are being damped by the shear force. As shown in equation 5.2, the shear force is proportional to the velocity, which means that the high velocity regions are damped out more than the low velocity regions. For narrow diffuser widths, addition of the wall-boundary layers has a stabilizing effect on the core flow, which is most likely due to the increased dissipation rate. For narrow diffusers, the effect of the wall-boundary layers results in a decrease of the critical inlet flow angle and the number of rotating cells. The propagation speed of the rotating cells remains unchanged. This is in good agreement with Abidogun [6] who found that compared to the influence of the diffuser radius ratio on the propagation speed, the influence of the diffuser width ratio is insignificant. For all impeller speeds, transition is found to be around $h/r_2 = 0.0047$. Because transition occurs at the same $h$ for all impeller speeds, the effect of the wall-boundary layers is found not to depend on $Re_h$.

It is important to note, that the obtained diffuser width ratio $h/r_2$ where transition takes place, has no indicative value for the transition in the real compressor configurations, because an overestimated friction force per unit volume is used. But this analysis shows that the wall boundary layers do have a significant influence on the two-dimensional core flow when the diffuser width becomes small enough.

Although the quasi-steady viscous approach helps to predict the influence of the wall boundary layers on the two-dimensional core flow, it does not take the retardation effect of the core-flow
$h/r = 1.32, \quad \Omega = 6.8^\circ$

The effect of the quasi-steady viscous contribution of the wall boundary layers, where solutions at $t = 0.6$ s are represented by contours of velocity magnitude, $u = \pm 1.52\cdot 10^6$, $\alpha_m = 6.8^\circ$.
motion with respect to the wall-boundary layers into account. When a fluid is driven by periodic changes in pressure, the wall-boundary layer responds immediately while the core flow needs some time to adapt to the new velocity due to the inertial effects. Therefore, a correction needs to be applied to the quasi-steady viscous contribution.

### 5.2.2 Corrected viscous contribution

The correction to the quasi-steady viscous contribution is applied according to the physical model developed by Meuleman [50] to study the unsteady internal flow driven by periodic changes in pressure. Here, the flow between two parallel plates of length $L$ is considered, where the distance between the plates is the channel width $h$. To determine the correction, the difference is considered between the exact solution of the unsteady momentum equation and a solution obtained by the quasi-steady approach for the viscous contribution. In appendix B the model is briefly described. For more details and for the exact derivation of the equations, the reader is referred to Meuleman [50].

According to Meuleman [50], the exact solution of the unsteady momentum equation can be expressed in terms of the mass flow:

$$\frac{d\dot{m}}{dt} = -a \frac{\Delta p}{L} + 2b \tau_w,$$  \hspace{1cm} (5.4)

where $\tau_w$ is the wall shear stress, $\Delta p$ the pressure difference across the channel length $L$ and $a = bh$ the area of the channel perpendicular to the flow. The quasi-steady momentum equation is formulated as follows:

$$\frac{d\dot{m}}{dt} = -a \frac{\Delta p}{L} + 2b \tau_{qs},$$  \hspace{1cm} (5.5)

where the quasi-steady wall shear stress for the fully-developed laminar flow case can be written as

$$\tau_{qs} = -\frac{6\nu}{ha} \dot{m}_{qs}.$$  \hspace{1cm} (5.6)

Since the correction $\chi$, which is made with the assumption of the quasi-steady viscous contribution, equals the difference between equation 5.4 and equation 5.5,

$$\chi = \tau_w - \tau_{qs},$$  \hspace{1cm} (5.7)

the quasi-steady momentum equation should then be improved with $\chi$ according to:

$$\frac{d\dot{m}}{dt} = -a \frac{\Delta p}{L} + 2b \{\tau_{qs} + \chi\}.$$  \hspace{1cm} (5.8)

To solve equation 5.8, the correction $\chi$ needs to be approximated. The following approximation was derived by Meuleman [50],

$$\chi_{app} = C \frac{h^2}{\nu} \frac{d\tau_{qs}}{d\dot{m}} \frac{d\dot{m}}{dt},$$  \hspace{1cm} (5.9)

where $C = 1/60$. The exact correction $\chi$ as well as the approximation of the correction $\chi_{app}$ are dependent on the dimensionless Stokes number, which characterizes the relative importance of the time-dependent inertia term with respect to the viscous term,

$$Stk = \frac{h}{2} \sqrt{\frac{n}{\nu}},$$  \hspace{1cm} (5.10)

where $n = 2\pi f$, with $f$ the frequency of the oscillation. Meuleman [50] has shown that the correction $\chi_{app}$ applies for $Stk < 2$, and that it should become smaller for $Stk > 2$ and
eventually become zero for large Stokes numbers. To make the approximated correction $\chi_{\text{app}}$ become smaller for large Stokes numbers and eventually become zero, a relaxation equation of this term is suggested according to

$$\tau \frac{d\chi}{dt} = C \frac{h^2}{\nu} \frac{d\dot{w}_{ws}}{d\dot{m}} \frac{d\dot{m}}{dt} - \chi,$$

(5.11)

which should be used in combination with equation 5.8. The time necessary for the core flow to adapt to the new velocity is defined by the chosen relaxation time scale $\tau$. The relaxation time scale $\tau$ is determined from the Stokes number in equation 5.10,

$$\tau = \frac{h^2}{2\nu \pi Stk^2},$$

(5.12)

Meuleman [50] has shown that this relaxation equation applies very well for time scales between the two extremes, corresponding to the Stokes numbers $2 < Stk < 200$. For larger time scales the old solution without correction is obtained, as in equation 5.5, and for smaller time scales the correction has a positive influence on the solution. Meuleman [50] has chosen $Stk = 20$ to be used for determination of the relaxation time scale, because it resulted in the best correction results.

Using equation 5.10, the Stokes number $Stk$ corresponding to the numerical model is estimated for the reference operating flow conditions, as given in table 2.1. For diffuser radius ratios $r_3/r_2 = 1.5 - 2$, the rotating stall cell frequencies vary between $f_{rs} = 28$ [Hz] and $f_{rs} = 15$ [Hz]. Taking these frequencies and different diffuser widths $h/r_2 = 0.025 - 0.1$ into consideration, the Stokes number values are found to be around $Stk \approx 50 - 200$. This means that the correction should be applied for this situation.

Since $Stk$ and $C$ are influencing the relaxation time, they are both varied to study the influence of the unsteady viscous contribution with respect to the quasi-steady viscous contribution. The Stokes number $Stk$ represents the frequency, and the value of $C$ represents the magnitude of $\chi_{\text{app}}$. The results obtained for different values of $Stk$ and $C$ are shown in figure 5.18, where $Stk$ is varied along the vertical axis and $C$ along the horizontal axis.

Figure 5.18 shows that for large disturbance frequencies $Stk$ and large values of $C$, where the quasi-steady viscous contribution is corrected with $\chi$, the rotating flow structures are being damped out by the retardation effect. On the other hand, for small values of $Stk$ and small values of $C$, the correction $\chi$ goes to zero and no damping of the flow structures is observed. For the two-dimensional rotating instability obtained with the numerical model $Stk \approx 50 - 200$. According to figure 5.18, at $Stk \approx 50 - 200$ damping of the cells due to the retardation effect occurs, and therefore this effect should be taken into consideration.

5.2.3 Discussion

The numerically obtained influence of the wall-boundary layers is compared with the observations found in the literature. The influence of the diffuser width on the vaneless diffuser performance is investigated experimentally by Abdelhamid and Bertrand [3], Dou [15] and Abidogun [5, 6], and theoretically by Dou and Mizuki [16], Senoo and Kinoshita [55] and Kinoshita and Senoo [34]. Tsurusaki et al. [60] has made some corrections to the method of Kinoshita and Senoo [34] and has accounted for the influence of the diffuser width.

Abdelhamid and Bertrand [3], Dou [15] and Dou and Mizuki [16] have found that the vaneless diffuser performance is different for narrow and wide vaneless diffusers. At higher diffuser width ratios Abdelhamid and Bertrand [3] have observed an intermittent onset of oscillations before
### 5.2. DIFFUSER WIDTH RATIO

<table>
<thead>
<tr>
<th>Stk</th>
<th>$C = 1/10000$</th>
<th>$C = 1/60$</th>
<th>$C = 1/20$</th>
<th>$C = 1$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.01</td>
<td><img src="image1" alt="Diagram" /></td>
<td><img src="image2" alt="Diagram" /></td>
<td><img src="image3" alt="Diagram" /></td>
<td><img src="image4" alt="Diagram" /></td>
</tr>
<tr>
<td>20</td>
<td><img src="image5" alt="Diagram" /></td>
<td><img src="image6" alt="Diagram" /></td>
<td><img src="image7" alt="Diagram" /></td>
<td><img src="image8" alt="Diagram" /></td>
</tr>
<tr>
<td>50</td>
<td><img src="image9" alt="Diagram" /></td>
<td><img src="image10" alt="Diagram" /></td>
<td><img src="image11" alt="Diagram" /></td>
<td><img src="image12" alt="Diagram" /></td>
</tr>
<tr>
<td>100</td>
<td><img src="image13" alt="Diagram" /></td>
<td><img src="image14" alt="Diagram" /></td>
<td><img src="image15" alt="Diagram" /></td>
<td><img src="image16" alt="Diagram" /></td>
</tr>
</tbody>
</table>

**Figure 5.18:** Effect of the unsteady viscous contribution, where solutions at $t = 0.6 \ [s]$ are represented by contours of velocity magnitude, $r_3/r_2 = 1.52$, $\alpha_m = 6.8^\circ$
the fully developed stall occurred, while at smaller width ratios they have observed an abrupt onset of oscillations, which was very well defined. Dou [15] has observed that the flow losses in the vaneless diffuser change with the variation of the diffuser width and the inlet flow angle. The wall-friction loss was found to be the primary loss in the narrow diffusers and a smaller part of the total loss in the wide diffusers. At small inlet flow angles, the loss in the diffuser consists primarily of the wall-friction loss and the secondary loss, while at large inlet flow angles, the loss in the diffuser is mainly composed of the wall-friction loss and the diffusion loss. Dou and Mizuki [16] have analyzed the vaneless diffuser flow with a three-dimensional wall-boundary-layer theory. They have found that an increase in radius ratio will result in a critical inlet flow angle increase if an inviscid core flow exists throughout the diffuser channel, which is in good agreement with the numerical model results. Besides, they also found that an increasing radius ratio will have no effect on the critical inlet flow angle if the inviscid core flow does not exist when separation occurs.

Senoo and Kinoshita [55], Kinoshita and Senoo [34], Abidogun [5, 6], Tsurusaki et al. [60] and Abdelhamid and Bertrand [3] have found that the critical inlet flow angle is dependent on the diffuser width ratio. Abdelhamid and Bertrand [3] and Abidogun [6] have both found that the critical flow coefficient at which the onset of oscillations was observed increased with decreasing diffuser width ratio $h/r_2$. Senoo et al. [56] and Senoo and Kinoshita [55] have developed a model to analyse the asymmetrical flow between the two diffuser walls, which they also have used to study the influence of the diffuser width ratio on the critical inlet flow angle. Senoo and Kinoshita [55] have found that the critical inlet flow angle increases with increasing diffuser width ratio, which is compared with the experimental data in Kinoshita and Senoo [34], and a good agreement is found. Their observation also agrees with the measurements of Abidogun [5] who has found that the critical inlet flow angle decreases as the diffuser width ratio is decreased for fixed diffuser radius ratios. Tsurusaki et al. [60] have measured the critical inlet flow angles with respect to the different diffuser width ratios and have made a comparison with the prediction curves of Kinoshita and Senoo [34]. They have used correlation fitted with their own experimental data to give an empirical formula for prediction of the critical inlet flow angles, showing that the critical inlet flow angle increases with increasing diffuser width.

The theory and measurements of Senoo and Kinoshita [55], Kinoshita and Senoo [34], Tsurusaki et al. [60] and Abidogun [5] show that the critical inlet flow angle increases with increasing diffuser width ratio. And the measurements of Abdelhamid and Bertrand [3] and Abidogun [6] show the opposite, which results in a decrease of the critical inlet flow angle with increasing diffuser width ratio. The exact explanation for this difference cannot be given, because different environments and operating flow conditions are used. It is not known if all the authors have used exactly the same way of measurement, by for example keeping the operating flow conditions unchanged after changing the diffuser width, or not. The difference in results might also be due to the different flow mechanisms causing the vaneless diffuser instability in different configurations, since the transition between the narrow and wide diffusers is not exactly defined.

Based on the data obtained by the current numerical analysis and the data found in the literature, the following conclusion can be drawn about the influence of the diffuser width. When the diffuser width is changed, there are two significant effects on the vaneless diffuser flow. First, change of the diffuser width has a direct effect on the frictional forces near the walls. As shown by the numerical model in this chapter, additional friction forces and their growth result in dissipation of the two-dimensional rotating structures, which means that there is less tendency for instability. Second, by contracting the diffuser, the flow is less inclined towards the tangential direction for the same mass flow rates, which also contributes to the stability of the two-dimensional core-flow region. But when the diffuser width changes from wide to narrow, the
structure of the flow throughout the diffuser channel can also change. Different flow structures can result in different types of flow mechanisms initiating the rotating stall instability. As many authors already suggested, it is probably the three-dimensional wall-boundary-layer instability that takes over in the narrow diffusers.

5.3 The impeller

Here, the number of jet-wakes prescribed at the diffuser inlet, representing the number of impeller blades, is varied. The influence of the number of impeller blades $N$ on the two-dimensional rotating instability is shown in figure 5.19, where the critical inlet flow angle, the number of rotating cells and their relative propagation speed are plotted versus the number of impeller blades. The number of impeller blades is chosen to be a prime, because it is common practice to use prime numbers for the impeller vanes of centrifugal compressors. For $N = 23$ the diffuser flow is found to be unstable at inlet flow angles between 0° and 20°, and therefore, no critical inlet flow angle is given at this value of $N$.

Figure 5.19 shows that the number of impeller blades influences the stability limit of the vaneless diffuser, but no particular trend of the critical inlet flow angle versus the number of impeller blades is found. The influence on the critical inlet flow angle indicates that the proper choice of the number of impeller blades can contribute to better stability of the vaneless diffuser core flow. The number of the impeller blades also has some influence on the number of rotating cells, but the number of cells stays close to the estimated maximum number of cells that fits into the given diffuser geometry. For the number of cells, the same trend is obtained as for the critical inlet flow angle, which might state that there is some link between the critical inlet flow angle and the number of rotating cells. Figure 5.19 also shows that the propagation speed of the two-dimensional rotating instability remains unchanged although the number of rotating cells changes with the varying number of impeller blades. Moreover the influence of the jet-wake shape and intensity on the core flow stability in the numerical model can be found in Ljevar et al. [41].

As shown earlier the influence of the $u_{jet}/u_{wake}$ ratio, the $\theta_{wake}/\theta_{jet}$ ratio, the jet-wake steepness $D$ and the additional velocity fluctuations on the critical inlet flow angle, number of rotating cells and their propagation speed is very small. This indicates that the shape and intensity of the jet-wake has a negligible influence on the characteristics of the two-dimensional rotating instability. This is in good agreement with the two-dimensional inviscid flow analysis of the vaneless diffuser flow, which is performed by Tsujimoto et al. [59]. As mentioned in chapter 4, they have studied the simplified case, where the effects of the impeller are not taken into account, since zero velocity fluctuations are prescribed at the diffuser inlet. Besides the simplified case, Tsujimoto et al. [59] have also studied the influence of the impeller on the vaneless diffuser flow. They have compared the simplified case with the two cases with modeled impeller flow with $r_1/r_2 = 0.5$, $\beta = 20^\circ$ and $\beta = 90^\circ$, where $r_1$ is the impeller inlet radius, $r_2$ the impeller outlet radius and $\beta$ the impeller vane angle. The results obtained by Tsujimoto et al. [59] show that the effects of the impeller are small and that the general tendencies as to $r_3/r_2$ and $m$ can be represented by the simplified case. According to Tsujimoto et al. [59], the negative slope of pressure performance and the inertia of the fluid between the impeller vanes affect the fluctuation of the flow through the impeller flow channel. These effects, along with inertia effects of the impeller upstream flow, are strong enough to suppress the velocity fluctuation at the diffuser inlet. According to Tsujimoto et al. [59], there are also experimental evidences showing that the vaneless diffuser rotating stall is unaffected by the upstream impeller.

In addition, also the influence of the impeller speed on the relative propagation speed of
rotating cells is investigated. Therefore, the vaneless diffuser core flow is brought into instability at three different impeller speeds. This is done by gradually decreasing the mass flow rate starting from the stable operating flow condition. In figure 5.20 the relative propagation speed of the cells is plotted versus the impeller tip speed for two diffuser radius ratios. The results show that the impeller tip speed does not influence the relative propagation speed of the two-dimensional rotating instability. This agrees well with the measurements of Abdelhamid and Bertrand [3] and Frigne and van den Braembussche [24] who made the same observation. Also the critical inlet flow angle and the number of rotating cells remain unchanged along the lines of constant diffuser radius ratio. Figure 5.20 also shows that the propagation speed of the cells is higher for $r_3/r_2 = 1.2$ than for $r_3/r_2 = 1.52$. Since the number of cells does not influence the propagation speed as mentioned in section 5.1.2, this difference is probably only due to a different distance of the cells from the impeller. For $r_3/r_2 = 1.2$, rotating cells are closer to the diffuser inlet where the tangential velocity component is higher, which results in a larger propagation speed of the cells.

5.4 Discussion

In this section, a few general remarks are made, which are drawn from the overall parameter analysis discussing the combined influence of multiple parameters.

It is shown in subsection 5.1.3 that differences between the two-dimensional rotating instabilities, corresponding to different diffuser radius ratios, can be noticed in the pressure fluctuations. It is known that the rotating stall pattern is not only influenced by the compressor configura-
5.4. DISCUSSION

Figure 5.20: Relative propagation speed of the cells versus the impeller tip speed, obtained with the two-dimensional core-flow model.

The propagation speed of rotating cells does not seem to be influenced by any other parameter than the diffuser radius ratio. This is in good agreement with measurements of Abidogun [6] who has found that the effect of the Reynolds number, number of impeller blades or diffuser width ratio on the propagation speed of the cells is insignificant compared to the effect of the diffuser radius ratio. It seems that only the size of the diffuser space influences the propagation speed of rotating cells. When the diffuser radius ratio increases, the number of cells and their propagation speed decrease but the size of the cells and their distance from the impeller increase. Since the
propagation speed did not change when the number of cells changed, it is probably the size of the rotating cells and/or their distance from the impeller that influence the propagation speed obtained by the two-dimensional numerical model. It is assumed that in this two-dimensional numerical model, where no influence of the wall-boundary layers is taken into account, the propagation speed is probably only influenced by the distance of the cells from the impeller. The critical inlet flow angle is mainly influenced by the diffuser radius ratio, while the number of the impeller blades can slightly change the value of the critical inlet flow angle corresponding to a given diffuser radius ratio. The number of cells and their size are determined by the vaneless diffuser space, and by the mass flow rate.

The fact that there are also publications suggesting that generation and characteristics of the rotating stall pattern depend on the coupling conditions between the impeller and diffuser, such as Abdelhamid [1], indicates that it is still not known in what way the impeller influences the vaneless diffuser flow and how large this effect is. According to the current numerical model, the shape and intensity of the jet-wake pattern have negligible influence on the stability limit and the two-dimensional rotating instability, while the effect of the number of impeller blades $N$ is slightly higher, but still not as large as that of the diffuser radius ratio.
Chapter 6

Concluding discussion

To increase the operating range of centrifugal compressors, more knowledge about rotating stall is required. This study is focused on rotating stall occurring in wide vaneless diffusers, where it can be assumed that mainly two-dimensional core flow is present. To study the two-dimensional core-flow instability within wide vaneless radial diffusers, a laminar incompressible flow model is developed within Fluent, and the results are validated with experiments. The numerical model is used to analyse physical aspects of the instability and to evaluate the influence of geometry parameters. Here, the main conclusions from the thesis are discussed, and some recommendations for the future research are suggested.

6.1 Conclusions

The two-dimensional rotating instability, associated with rotating stall in wide vaneless diffusers, is found to exist in the two-dimensional numerical model of the vaneless diffuser core flow. It is found to be similar to rotating stall since it develops within a few impeller revolutions, and it consists of a number of rotating cells that propagate with a fraction of impeller speed around the circumference. The flow field analysis showed that the cells rotate around their own center in the opposite rotation direction than the impeller. It is shown that the stability limit of the two-dimensional core flow can be expressed in terms of the critical flow angle at the diffuser inlet. This agrees well with the literature where often the same stability criterion is used for rotating stall.

The numerical model results are compared with experimental data found in the literature and with the measurements performed in two experimental setups. When compared with experimental data from the literature, similar arrangement of the flow structures is found within the velocity and pressure fields. To reconstruct the two-dimensional rotating instability obtained with the numerical model, a water model of the wide vaneless diffuser is build. It is shown that in the $r - \theta$ plane vortices occur at the shear-layer between the impeller and the slow moving fluid near the outlet. The flow field obtained by the current numerical model is also compared with the flow field obtained within the wide vaneless diffuser behind the radial flow pump impeller. Since the wide vaneless diffuser behind the radial flow pump impeller has no volute at the diffuser outlet, the rotating stall cells are free to move outside the vaneless diffuser space. The outlet boundary condition in this experimental setup is not affecting the flow exactly at the diffuser outlet, but slightly outside the vaneless diffuser space. Therefore, better agreement is found with the numerical model having larger diffuser radius ratio than the design radius ratio. Similar flow structures are obtained having almost the same size, propagation speed and
distance from the impeller as obtained in the numerical model. The difference in the critical flow angle and the number of cells is ascribed to the different effect of the outlet boundary condition. The slight differences in the number of cells and their vorticity are ascribed to the absence of the dissipation effects in the current numerical model, which are generated by the turbulent viscosity and the wall-boundary layers. It is shown that these effects will lead to a decrease in the number of rotating cells, but the exact quantity of these effects could not be modeled. Alikeness of the numerical model results with the experimentally observed rotating stall, and with the experimental data found in the literature, implies that the two-dimensional core-flow instability might contribute to rotating stall in the wide vaneless diffusers.

To study the origin of the two-dimensional core-flow instability, a physical analysis of the instability is performed. The two-dimensional inviscid stability analysis performed by Tsujimoto et al. [59] shows that the two-dimensional rotating flow instability is inviscid. The two-dimensional inviscid stability analysis as well as the current numerical model show that the two-dimensional core flow instability is not a jet-wake instability. They both show that the instability associated with vaneless diffuser rotating stall occurs even with uniform outward flow. With the current numerical model it is also shown that the jet-wake shape and intensity have a negligible influence on the two-dimensional rotating instability. Furthermore, it is shown that the instability will not occur, zero flow is prescribed at the inlet. This might indicate that the tangential acceleration by itself can not be responsible for the occurrence of the two-dimensional rotating instability on the short time scale, unless the instability generation is inhibited by the numerical artefact. Furthermore, it is shown that the tangential forcing plays a significant role in the origin of the core-flow instability. The ratio between the tangential forcing and convective forces has to be larger than a certain value, which is in this case $1/\tan \alpha_{cr}$, in order to prevent instability. The instability is expected when $Re < \tan (\alpha_{cr}) \sqrt{Ta}$. Since the jet-wake pattern is not responsible for the occurrence of the two-dimensional rotating instability, it is appropriate to assume that primarily the shear-layer between the impeller region and the diffuser outlet, together with tangential acceleration, are responsible for the core-flow instability. Although the jet-wake and/or the tangential forcing are not found to be directly responsible for the occurrence of the two-dimensional rotating instability, they may have some influence on the stability limit or the instability characteristics. By varying the impeller startup condition, it is shown that the instability occurs only when it is triggered by the high discontinuity in the tangential velocity component. If the impeller is slowly accelerated, no circumferential disturbances and thus no instability will occur. It is found that for Strouhal numbers, based on the impeller rotation frequency, above a certain value ($St > 0.24$) large-scale instabilities are to be expected.

It is also shown that available engineering turbulence models in Fluent, damp the large flow structures such as the rotating cells. This is ascribed to the excessive viscous dissipation rate. The engineering turbulence models overestimate the amount of the turbulent viscosity, while in the current numerical model it is underestimated. In the numerical model, the additional dissipation effect due to turbulent viscosity and wall-boundary layers is not taken into account. Therefore, it is assumed that in practice most likely a compromise is found between these two situations. It is shown that the same effect is obtained when the molecular viscosity is increased or the Reynolds number at the diffuser inlet is decreased in the current numerical model. The additional viscosity dissipation rate is found to slightly decrease the number of rotating cells, and to have a stabilizing effect on the core flow, which leads to slightly lower critical flow angles. Since the viscous dissipation rate in the current numerical model is underestimated, the recovery or the transition from the unstable to stable operating flow condition could not be obtained.

To study the characteristics of the two-dimensional rotating instability, the influence of the geometry parameters is also investigated. The diffuser radius ratio is found to have a significant
6.2. DISCUSSION AND RECOMMENDATIONS

Influence on the critical inlet flow angle, the number of rotating cells and their propagation speed. As the diffuser radius ratio decreases, the critical inlet flow angle also decreases, while the number of rotating cells as well as the propagation speed of the cells both decrease. The influence of the diffuser radius ratio obtained by the numerical model is compared with the two-dimensional inviscid stability analysis performed by Tsujimoto et al. [59] and with the experimental data found in the literature, and generally a good agreement is found. It is shown that the maximum number of rotating cells, occurring just after the stability limit is exceeded, can be estimated based on the diffuser geometry. It is found that not only the diffuser geometry, but also the operating flow conditions influence the flow characteristics. The number of rotating cells is not only determined by the diffuser geometry, but also by the mass flow rate. As the mass flow rate decreases the number of cells is found to slightly decrease. The propagation speed of rotating cells does not seem to be influenced by any other parameter but the diffuser radius ratio. It is shown that the propagation speed is not influenced by the number and size of the rotating cells, but by the distance of the cells from the impeller. Since the pressure fluctuations contain information about the flow structures and their flow dynamics, the relation between the pressure signals and the flow characteristics is found to be very useful for the interpretation of the measured pressure signals during compressor operation. The influence of the diffuser width and the wall-boundary layers is qualitatively predicted by adding a quasi-steady viscous contribution to the numerical model. The quantity of this effect could not be obtained with the current numerical model. It is shown that for wide vaneless diffusers, the wall-boundary layers do not contribute to the effective dissipation of the rotating structures. In case of wide diffusers, influence of the wall-boundary layers on the vaneless diffuser core-flow is according to the numerical model quite slight. For narrow diffusers, addition of the wall-boundary layers has a stabilizing effect on the core flow, which is most likely due to the increased dissipation rate. The critical inlet flow angle and the number of rotating cells decrease, while the propagation speed of the rotating cells remains unchanged. It is also shown, by applying the unsteady viscous contribution, that the core-flow structures are being damped when the relaxation time-scales become small. The number of impeller blades \( N \) is found to slightly influence the two-dimensional rotating instability, but this effect is not as large as that of the diffuser radius ratio. Slight influence of \( N \) on the critical inlet flow angle indicates that the proper choice of the number of impeller blades can contribute to better stability of the vaneless diffuser core flow. By prescribing the fixed flow conditions at the diffuser inlet, interaction between the impeller and diffuser flow is not taken into account in the current numerical model. Since it is not known if the upstream impeller affects the generation and characteristics of rotating stall in the vaneless diffusers, it is not known whether the model deviates much from practice or not. The current numerical model and the linear stability analysis performed by Tsujimoto et al. [59] show that the vaneless diffuser rotating stall is nearly unaffected by the upstream impeller flow, while the model of Abdelhamid [1] shows that the rotating stall pattern depends on the coupling conditions between the impeller and diffuser.

6.2 Discussion and recommendations

When comparing the flow dynamics of the two-dimensional core-flow instability obtained with the current numerical model to the real vaneless diffuser flow, some issues must be taken into account. Therefore, a discussion is held about the values and shortcomings of the two-dimensional numerical model, with emphasis on its purpose to review the basic flow dynamics of the rotating stall phenomenon in wide vaneless diffusers.

Since the vaneless diffuser flow is three-dimensional, the applied two-dimensional approach
has its limitations. One of the limitations of the two-dimensional modeling is, that any three-dimensional flow mechanisms that might be responsible for the occurrence of rotating stall, besides the two-dimensional core-flow instability, cannot be detected. Another limitation is that the exact influence of the wall-boundary layers on the two-dimensional core flow instability cannot be determined. Furthermore, the current numerical model probably overestimates the dissipation due to the turbulent viscosity. Therefore, other schemes could be used in order to improve modeling of the turbulent dissipation.

In the real compressor configurations, the vaneless diffuser outlet is connected to a volute, where the flow is highly three-dimensional and most likely swirling. Since the vaneless diffuser core-flow model is two-dimensional, constant static pressure is prescribed at the diffuser outlet. This condition might approach the effect of the volute flow on the diffuser, but the presence of the nearby wall of the volute, should also be taken into consideration. It is shown that the influence of the nearby wall can contribute to the dissipation of the rotating stall cells. Unlike in the current numerical model, in the real compressor stage the vaneless diffuser flow and the volute wall are separated by the three-dimensional and swirling flow through the volute. Therefore, it is assumed that the effect of the volute wall mainly influences the volute flow itself and that where the wall is close to the diffuser, which is a small region, the dissipation effect due to the nearby wall is small. It is also assumed that the constant pressure boundary condition approaches the effect of the volute on the diffuser flow, but it might also use additional effects such as the influence of the wall, swirl or asymmetry. The exact influence of the volute is not known, and therefore this uncertainty should be taken into consideration.

It is not possible to model any compressibility effects with the current numerical model. In the real compressor configurations $Ma \approx 0.6$, which means that the flow is subsonic, $Ma < 1$. In the flow of gases, compressibility effects due to variations in the density may still be neglected as long as the velocity is not too high. For low to medium Mach numbers $0.3 < Ma < 0.8$, density changes can be considerable, but the gas dynamic phenomena like shock waves do not have to be considered. On the other hand, even at lower Mach numbers, the geometry may be shaped such that locally $\Delta p$ can become large enough so that compressibility effects do affect the flow compressibility. It is assumed that the density changes due to geometry are relatively small and that the compressibility effects are not the main cause cause of the rotating stall instability. It is not proven that acoustics-waves generated at medium Mach numbers $0.3 < Ma < 0.6$, can influence the rotating stall phenomenon, but additional research is worthwhile. It is known that rotating stall does occur in incompressible flows since it also occurs in radial flow pumps.

It is shown that the current numerical model can predict most of the general characteristics of rotating stall, which are usually obtained in the experiments, namely the critical inlet flow angle, the number of rotating cells, the propagation speed of the cells and the pressure fluctuation fields. These calculations are two-dimensional, frictionless and can not show the real, viscous, compressible and three-dimensional flow. Despite this disadvantage, the procedures show the global influence of the single parameters and are helpful for interpretation of the experimental results, and more importantly for interpretation of the core-flow instability and the rotating stall flow dynamics. In order to have a more detailed look on the vaneless diffuser flow, including the influence of the secondary flows due to the wall-boundary layers, the two-dimensional numerical model could be extended into the three-dimensional model of the vaneless diffuser flow. Then, also better outlet boundary condition could be used, since the volute can be added to the three-dimensional model. To investigate the interaction between the impeller and diffuser flow, the impeller could then also be modeled in front of the diffuser inlet.
Bibliography


### Symbols

<table>
<thead>
<tr>
<th>Latin letters</th>
<th>Symbol</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>surface area</td>
<td>$a$</td>
<td></td>
<td>[m$^2$]</td>
</tr>
<tr>
<td>amplitude</td>
<td>$A$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>channel width</td>
<td>$b$</td>
<td></td>
<td>[m]</td>
</tr>
<tr>
<td>relaxation constant</td>
<td>$C$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>diameter</td>
<td>$d$</td>
<td></td>
<td>[m]</td>
</tr>
<tr>
<td>difference between outer and inner radius</td>
<td>$d_0$</td>
<td></td>
<td>[m]</td>
</tr>
<tr>
<td>constant for jet-wake steepness</td>
<td>$D$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>frequency</td>
<td>$f$</td>
<td></td>
<td>[Hz]</td>
</tr>
<tr>
<td>diffuser width</td>
<td>$h$</td>
<td></td>
<td>[m]</td>
</tr>
<tr>
<td>imaginary unit</td>
<td>$j$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>constant</td>
<td>$K$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>distance between two rotating cells</td>
<td>$l$</td>
<td></td>
<td>[m]</td>
</tr>
<tr>
<td>channel length</td>
<td>$L$</td>
<td></td>
<td>[m]</td>
</tr>
<tr>
<td>number of rotating cells</td>
<td>$m$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>mass flow rate</td>
<td>$\dot{m}$</td>
<td></td>
<td>[kg/s]</td>
</tr>
<tr>
<td>Mach number</td>
<td>$Ma$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>frequency</td>
<td>$n$</td>
<td></td>
<td>[Hz]</td>
</tr>
<tr>
<td>number of jet-wakes or impeller blades</td>
<td>$N$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>pressure</td>
<td>$p$</td>
<td></td>
<td>[Pa]</td>
</tr>
<tr>
<td>stagnation pressure</td>
<td>$p_0$</td>
<td></td>
<td>[Pa]</td>
</tr>
<tr>
<td>static pressure</td>
<td>$p_{st}$</td>
<td></td>
<td>[Pa]</td>
</tr>
<tr>
<td>volume flow rate per m</td>
<td>$Q$</td>
<td></td>
<td>[m$^2$/s]</td>
</tr>
<tr>
<td>radial position</td>
<td>$r$</td>
<td></td>
<td>[m]</td>
</tr>
<tr>
<td>number of impeller revolutions</td>
<td>$R$</td>
<td></td>
<td>[rev]</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$Re$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>friction force per unit volume</td>
<td>$S$</td>
<td></td>
<td>[N/m$^3$]</td>
</tr>
<tr>
<td>Strouhal number</td>
<td>$St$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>Stokes number</td>
<td>$Stk$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>time</td>
<td>$t$</td>
<td></td>
<td>[s]</td>
</tr>
<tr>
<td>time necessary to cross the grid cell</td>
<td>$t_{cell}$</td>
<td></td>
<td>[s]</td>
</tr>
<tr>
<td>Taylor number</td>
<td>$Ta$</td>
<td></td>
<td>[−]</td>
</tr>
<tr>
<td>radial velocity component</td>
<td>$u$</td>
<td></td>
<td>[m/s]</td>
</tr>
<tr>
<td>impeller tip speed</td>
<td>$U_2$</td>
<td></td>
<td>[m/s]</td>
</tr>
<tr>
<td>mean stream velocity</td>
<td>$U_{\infty}$</td>
<td></td>
<td>[m/s]</td>
</tr>
</tbody>
</table>

*Continued on next page*
<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>( v )</td>
<td>tangential velocity component</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( v_r )</td>
<td>radial velocity component</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( v_\theta )</td>
<td>tangential velocity component</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( V )</td>
<td>velocity magnitude</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( V_r )</td>
<td>radial component of the steady flow</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( V_\theta )</td>
<td>tangential component of the steady flow</td>
<td>([\text{m/s}])</td>
</tr>
<tr>
<td>( z )</td>
<td>z direction</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( x )</td>
<td>x direction</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( y )</td>
<td>y direction</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( Y )</td>
<td>sinusoidal function defined in eq. 2.11</td>
<td>([-\text{]})</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Greek letters</th>
<th>Definition</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>flow angle at diffuser inlet</td>
<td>([\text{o}])</td>
</tr>
<tr>
<td>( \beta )</td>
<td>vane angle</td>
<td>([\text{o}])</td>
</tr>
<tr>
<td>( \Gamma )</td>
<td>circulation</td>
<td>([\text{m}^2/\text{s}])</td>
</tr>
<tr>
<td>( \delta )</td>
<td>wall boundary layer thickness</td>
<td>([\text{m}])</td>
</tr>
<tr>
<td>( \epsilon )</td>
<td>turbulence dissipation rate</td>
<td>([\text{m}^2/\text{s}^3])</td>
</tr>
<tr>
<td>( \zeta )</td>
<td>vorticity</td>
<td>([\text{s}^{-1}])</td>
</tr>
<tr>
<td>( \theta )</td>
<td>circumferential position</td>
<td>([\text{o}])</td>
</tr>
<tr>
<td>( \theta_s )</td>
<td>distance between two sensors</td>
<td>([\text{o}])</td>
</tr>
<tr>
<td>( \mu )</td>
<td>dynamic viscosity</td>
<td>([\text{kgm}^{-1}\text{s}^{-1}])</td>
</tr>
<tr>
<td>( \mu_t )</td>
<td>turbulent or eddy viscosity</td>
<td>([\text{kgm}^{-1}\text{s}^{-1}])</td>
</tr>
<tr>
<td>( \nu )</td>
<td>kinematic viscosity</td>
<td>([\text{m}^2/\text{s}])</td>
</tr>
<tr>
<td>( \rho )</td>
<td>density</td>
<td>([\text{kg/mol}^3])</td>
</tr>
<tr>
<td>( \sigma )</td>
<td>slip factor</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( \sigma_0 )</td>
<td>slip correction factor</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( \tau )</td>
<td>relaxation time scale</td>
<td>([\text{s}])</td>
</tr>
<tr>
<td>( \tau_w )</td>
<td>wall shear stress</td>
<td>([\text{N/m}^2])</td>
</tr>
<tr>
<td>( \tau_{w qs} )</td>
<td>quasi-steady wall shear stress</td>
<td>([\text{N/m}^2])</td>
</tr>
<tr>
<td>( \phi )</td>
<td>flow coefficient</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( \varphi )</td>
<td>phase shift</td>
<td>([\text{o}])</td>
</tr>
<tr>
<td>( \chi )</td>
<td>correction</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( \chi_{\text{app}} )</td>
<td>approximated correction</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( \psi )</td>
<td>pressure coefficient</td>
<td>([-\text{]})</td>
</tr>
<tr>
<td>( \Omega )</td>
<td>angular impeller speed</td>
<td>([\text{rpm}])</td>
</tr>
<tr>
<td>( \omega_i )</td>
<td>angular impeller speed</td>
<td>([\text{rad/s}])</td>
</tr>
<tr>
<td>( \omega )</td>
<td>angular speed</td>
<td>([\text{rad/s}])</td>
</tr>
</tbody>
</table>
## Symbols and Signs

<table>
<thead>
<tr>
<th>Sign</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>cr</td>
<td>critical</td>
</tr>
<tr>
<td>i</td>
<td>impeller</td>
</tr>
<tr>
<td>im</td>
<td>imaginary</td>
</tr>
<tr>
<td>jet</td>
<td>jet region of the jet-wake pattern</td>
</tr>
<tr>
<td>m</td>
<td>mean</td>
</tr>
<tr>
<td>p</td>
<td>pressure signal</td>
</tr>
<tr>
<td>p1</td>
<td>pressure measurement location 1</td>
</tr>
<tr>
<td>p2</td>
<td>pressure measurement location 2</td>
</tr>
<tr>
<td>p3</td>
<td>pressure measurement location 3</td>
</tr>
<tr>
<td>r</td>
<td>real</td>
</tr>
<tr>
<td>rs</td>
<td>rotating stall</td>
</tr>
<tr>
<td>s</td>
<td>flow structure</td>
</tr>
<tr>
<td>st</td>
<td>stable</td>
</tr>
<tr>
<td>w</td>
<td>wall</td>
</tr>
<tr>
<td>wake</td>
<td>wake region of the jet-wake pattern</td>
</tr>
<tr>
<td>1</td>
<td>impeller inlet</td>
</tr>
<tr>
<td>2</td>
<td>impeller exit or diffuser inlet</td>
</tr>
<tr>
<td>3</td>
<td>diffuser exit</td>
</tr>
<tr>
<td>4</td>
<td>wall cylindrical tank</td>
</tr>
<tr>
<td>-</td>
<td>disturbance</td>
</tr>
<tr>
<td>Δ</td>
<td>step / difference</td>
</tr>
</tbody>
</table>
Appendix A

Two-dimensional inviscid flow model

In this appendix, the two-dimensional inviscid flow model as used in chapter 4 and 5 is shortly described. Here, the set of equations is given, which is solved to obtain the results presented in chapter 5. For more details about the model, the reader is referred to Tsujimoto et al. [59] and Tsujimoto and Acosta [58].

A.1 Flow in the vaneless diffuser

Consider a two-dimensional vaneless diffuser space with diffuser inlet radius $r_2$ and diffuser outlet radius $r_3$, as shown in figure A.1. The impeller is modeled to rotate with an angular velocity $\Omega$, and it has an infinite number of vanes with vane angle $\beta$. The flow is assumed to be inviscid and incompressible, and it is assumed that the disturbance is so small that a linear analysis is allowed. The unsteady components are represented by

$$v_r = \tilde{v}_r(r) \exp\{j (\omega_i t - m \theta)\} \tag{A.1}$$
$$v_\theta = \tilde{v}_\theta(r) \exp\{j (\omega_i t - m \theta)\}$$

where $j$ is the imaginary unit with respect to time $t$, $\omega_i$ is the angular frequency and $m$ is the number of stall cells. It is assumed that the steady flow in the vaneless diffuser space is characterized by

$$(V_r, V_\theta) = \left(\frac{Q}{2 \pi r}, \frac{\Gamma}{2 \pi r}\right) \tag{A.2}$$

where $V_r$ is the purely radial velocity, $Q$ volume flow rate per meter, and $V_\theta$ represents a potential vortex of net circulation $\Gamma$. In order to determine the unsteady flow components in the vaneless diffuser space due to vorticity, the vorticity transport equation is linearized and solved. The linearized vorticity transport equation can be written as

$$\frac{\partial \zeta}{\partial t} + V_r \frac{\partial \zeta}{\partial r} + V_\theta \frac{\partial \zeta}{r \partial \theta} = 0 \tag{A.3}$$

where $\zeta$ is the vorticity. According to Tsujimoto et al. [59], the linearized vorticity equation has a solution represented by

$$\zeta = \tilde{\zeta}_2 \exp \left\{ \pm j \omega_i \left[ t - \frac{\pi}{Q} (r_2^2 - r_2) \right] + j m \left[ \theta - \frac{\Gamma}{Q} \log \frac{r}{r_2} \right] \right\} \tag{A.4}$$

where

$$\Gamma \equiv 2 \pi r_2 \left\{ r_2 \Omega - \left( \frac{Q}{2 \pi r_2} \right) \cot \beta \right\} \tag{A.5}$$
It is also assumed that the relative flow exits the impeller tangentially to the vanes, and that the following condition needs to be satisfied at the diffuser inlet,

$$\tilde{v}_\theta = -\tilde{v}_r \cot \beta$$  \hspace{1cm} (A.6)

In order to obtain the unsteady flow components that have the vorticity of equation A.4 and at the same time satisfy the boundary condition in equation A.6, two potential flow components are added to the velocity field induced by vorticity of equation A.4. Then, the unsteady velocity field in the vaneless diffuser space is obtained such, to have the vorticity of equation A.4, and to satisfy the boundary condition in equation A.6. The velocity disturbance in the vaneless diffuser space can be represented by,

$$\tilde{v}_r = \tilde{\zeta}_2 \left\{ (R_R - jR_I) - \left( \frac{r_2}{r} \right)^{m+1} e^{-2j\beta} (R_R - jR_I) r_2 \right\} + \tilde{A} \left\{ \left( \frac{r}{r_2} \right)^{m-1} - \left( \frac{r_2}{r} \right)^{m+1} e^{-2j\beta} \right\}$$  \hspace{1cm} (A.7)

$$\tilde{v}_\theta = \tilde{\zeta}_2 \left\{ (\Theta_R - j\Theta_I) - j \left( \frac{r_2}{r} \right)^{m+1} e^{-2j\beta} (R_R - jR_I) r_2 \right\} - j\tilde{A} \left\{ \left( \frac{r}{r_2} \right)^{m-1} + \left( \frac{r_2}{r} \right)^{m+1} e^{-2j\beta} \right\}$$

where $\tilde{\zeta}_2$ and $\tilde{A}$ are the unknown complex constants, which can be determined from the boundary conditions described in the next sections, and where

$$R_R - jR_I = -j (F(r_0) + G(r_0))$$  \hspace{1cm} (A.8)

$$\Theta_R - j\Theta_I = F(r_0) - G(r_0)$$
and

\[
F(r_0) = \frac{1}{2} \int_{r_2}^{r_0} \exp \left\{ j \left[ \pm \frac{\omega_i}{Q} \left( r_1^2 - r_2^2 \right) - m \frac{\Gamma}{Q} \log \left( r_0 \right) \right]^m \right\} \left( \frac{r_0}{r_2} \right)^{m+1} dr_0
\]

\[
G(r_0) = \frac{1}{2} \int_{r_0}^{r_3} \exp \left\{ j \left[ \pm \frac{\omega_i}{Q} \left( r_1^2 - r_2^2 \right) - m \frac{\Gamma}{Q} \log \left( r_0 \right) \right]^m \right\} \left( \frac{r_0}{r_1} \right)^{m-1} dr_0
\]  

(A.9)

A.2 Boundary condition at diffuser exit

At the diffuser outlet it is assumed that the flow is discharged into the space with constant pressure, and therefore

\[
\left( \frac{\partial p}{\partial \theta} \right)_{r = r_3} = 0.
\]  

(A.10)

which applies to the \( \theta \) component of the momentum equation. The linearized \( \theta \) momentum equation is then given by

\[
\frac{\partial v_\theta}{\partial t} + V_r \frac{\partial v_\theta}{\partial r} + v_r \frac{\partial V_\theta}{\partial r} + v_\theta \frac{\partial v_r}{\partial \theta} + v_r v_\theta \frac{\partial V_r}{\partial \theta} + V_r v_\theta \frac{1}{r} = - \frac{1}{\rho r} \frac{\partial p}{\partial \theta}
\]  

(A.11)

The unsteady velocity components of the linearized \( \theta \) momentum equation can be obtained by substitution of velocities in equation A.7 into the equation A.1. Subsequently, substitution of the unsteady velocity components and the boundary condition A.10 into the linearized \( \theta \) momentum equation, will lead to the following relation that applies at the diffuser exit,

\[
P_\zeta \tilde{\zeta}_2 + P_A \tilde{A} = 0,
\]  

(A.12)

where

\[
P_\zeta = \left\{ j \omega_i - \frac{j m \Gamma}{2 \pi r_3^2} + \frac{Q}{2 \pi r_3^2} \right\} \dot{Z}_{\theta_3} + \frac{Q}{2 \pi r_3} Z_a
\]

\[
P_A = \left\{ j \omega_i - \frac{j m \Gamma}{2 \pi r_3^2} + \frac{Q}{2 \pi r_3^2} \right\} \dot{A}_{\theta_3} - \frac{Q}{2 \pi r_3} Z_b
\]

\[
Z_{\theta_3} = (\Theta r - j \Theta I) - j \left( \frac{r_2}{r_3} \right)^{m+1} e^{-2j\beta} (R_R - j R_I)_{r_2}
\]

\[
A_{\theta_3} = -j \left\{ \left( \frac{r_3}{r_2} \right)^{m-1} - \left( \frac{r_2}{r_3} \right)^{m+1} e^{-2j\beta} \right\}
\]

\[
Z_a = \exp \left\{ -j \left[ \frac{\omega_i}{Q} \left( r_3^2 - r_2^2 \right) - m \frac{\Gamma}{Q} \log \left( r_3 \right) \right]^m \right\} - \frac{m+1}{r_3} F(r_3) + j (m+1) \frac{r_2^{m+1}}{r_3^{m+2}} e^{-2j\beta} (R_R - j R_I)_{r_2}
\]

\[
Z_b = (m - 1) \frac{r_3^{m-2}}{r_2^{m-1}} - (m + 1) \frac{r_2^{m+1}}{r_3^{m+2}} e^{-2j\beta}
\]
A.3 Simplified case

In the model of Tsujimoto et al. [59], a distinction is made between the general and the simplified case. In the general case the modeled impeller flow is combined with the vaneless diffuser flow, while in the simplified case no impeller flow is taken into account. The simplified case with the following boundary conditions is considered

\[ \tilde{p}_3 = 0. \]  

(A.13)

\[ \tilde{v}_{r_2} = \tilde{v}_{\theta_2} = 0 \]  

(A.14)

The boundary condition in equation A.13 is already represented by equation A.12, and the boundary condition in equation A.14 can be represented by

\[ \tilde{v}_{r_2} = Z_{r_2} \tilde{\zeta}_2 + A_{r_2} \tilde{A}_2 = 0, \]  

(A.15)

where

\[ Z_{r_2} = \left( 1 - e^{-2j\beta} \right) (R_R - jR_I)_{r_2} \]  

(A.16)

\[ A_{r_2} = \left( 1 - e^{-2j\beta} \right) \]

From equations A.12 and A.15 the condition for the existence of a non-trivial solution can be obtained as follows,

\[ P_\zeta A_{r_2} - P_A Z_{r_2} = 0 \]  

(A.17)

Once the diffuser radius ratio \( r_3/r_2 \) and the number of cells \( m \) is given, the flow angle \( \alpha \) and the angular frequency \( \omega_i \) can be determined so that the real and imaginary parts of equation A.17 are satisfied. By considering \( \omega_i \) to be a complex number,

\[ \omega_i = \omega_r + j \omega_{im}, \]  

(A.18)

it can be shown that the disturbance is amplifying in the region where the flow angle \( \alpha \) is smaller than the determined critical flow angle \( \alpha_{cr} \). The Matlab function \textit{fminsearch} is used to find the radial velocity at the diffuser inlet \( V_{r_2} \) when \( \omega_{im} \) approaches zero. When \( \omega_{im} < 0 \) damping occurs and the solution is stable, and when \( \omega_{im} > 0 \) amplification occurs the solution is unstable. Since,

\[ \alpha = \arctan \left( \frac{V_r}{V_\theta} \right) = \arctan \left( \frac{Q}{\Gamma} \right) \]  

(A.19)

the critical flow angle \( \alpha_{cr} \) is determined using equation A.5 and

\[ Q = V_{r_2} \left( \frac{2 \pi r_2}{m V_{\theta_2}} \right). \]  

(A.20)

Since the corresponding real value of the complex number \( \omega_i \) is equivalent to the angular frequency, the propagation speed of the disturbance can be determined by

\[ V_p/V_{\theta_2} = \left( \frac{r_2 \omega_i}{m V_{\theta_2}} \right). \]  

(A.21)

By substitution of the flow tangency condition A.6 and the simplified boundary condition A.14 into equation A.17, it can be shown that the simplified flow is completely independent on the impeller.
Appendix B

Unsteady fully developed flow model

In this appendix, the physical model developed by Meuleman [50] is described that represents the unsteady compressor performance. The model is made by considering the difference between the exact solution of the unsteady momentum equation and a solution obtained by the quasi-steady approach of the viscous contribution. For more details and for the exact derivation of the equations, the reader is referred to Meuleman [50].

B.1 Exact solution of an oscillating channel flow

The channel, shown in figure B.1, is assumed to be infinitely long. This leads to a fully developed incompressible flow, which motion is independent of $x$. From the incompressible Navier-Stokes equations a linear equation is obtained,

$$\frac{\partial u}{\partial t} = - \frac{1}{\rho} \frac{dp}{dx} + \nu \frac{\partial^2 u}{\partial y^2}, \quad (B.1)$$

with boundary conditions $u = 0$ at $y = \pm h/2$. The pressure gradient is assumed to be set by a harmonically moving piston as

$$- \frac{1}{\rho} \frac{dp}{dx} = K \sin (n t), \quad (B.2)$$

Figure B.1: Illustration of an infinitely long channel
APPENDIX B. UNSTEADY FULLY DEVELOPED FLOW MODEL

where $K$ is a constant, and $n = 2 \pi f$, with $f$ the frequency of the oscillation. By solving the linear differential equation the exact unsteady velocity profile is obtained,

\[
  u(y,t) = -\frac{K}{n} \left\{ 1 - \frac{\cos \left( Stk \sqrt{\frac{y}{h^2}} \right)}{\cos (Stk)} \right\} \exp^{i n t}, \tag{B.3}
\]

where

\[
  Stk = Stk \sqrt{-i}, \tag{B.4}
\]

and the dimensionless $Stk$

\[
  Stk = \frac{h}{2} \sqrt{\frac{n}{\nu}}, \tag{B.5}
\]

is a Stokes number, that characterizes the relative importance of the time-dependent inertia term with respect to the viscous term. The exact time-dependent mass flow is obtained by multiplying the velocity profile by the density $\rho$ and integrating it over the height and width of the channel. The momentum equation B.1 can be integrated such that it gives an equation for the derivative of the mass flow,

\[
  \frac{d\dot{m}}{dt} = -a \frac{dp}{dx} + 2 b \rho \nu \frac{\partial u}{\partial y} \bigg|_w, \tag{B.6}
\]

where $\rho \nu \partial u/\partial y \bigg|_w = \tau_w$ is the wall-shear stress, with $\partial u/\partial y \bigg|_w$ the derivative of the velocity at the wall, and $a = bh$ the area of the channel perpendicular to the flow. Integration of equation B.6 over a channel length $L$ in $x-$ direction gives the exact momentum equation expressed in terms of the mass flow,

\[
  \frac{d\dot{m}}{dt} = -a \frac{\Delta p}{L} + 2 b \tau_w, \tag{B.7}
\]

where $\Delta p$ is the pressure difference across the channel length $L$.

**B.2 Quasi-steady approach of the viscous contribution**

For very small values of $n$, i.e., low frequency motions, the quasi-steady profile is obtained by expanding $\cos (\lambda) = 1 - \lambda^2/2! + ...$ in equation B.3. Meuleman [50] has shown that for slow oscillations the velocity distributions has the same phase as the pressure gradient that forces it. Furthermore, the velocity distribution is parabolic as in the steady flow case. According to Meuleman [50] the quasi-steady wall-shear stress $\tau_{wqs}$ can be written as a function of the mass flow,

\[
  \tau_{wqs} = -\frac{6 \nu}{h a} \dot{m}_{qs}. \tag{B.8}
\]

For this fully-developed laminar flow case the quasi-steady wall shear stress is found to depend linearly on the quasi-steady mass flow. Then, the quasi-steady momentum equation becomes,

\[
  \frac{d\dot{m}}{dt} = -a \frac{\Delta p}{L} + 2 b \tau_{wqs}. \tag{B.9}
\]
B.3 Determination of the correction term

Since the error $\chi$ made with assumption of a quasi-steady viscous term equals the difference between equation B.7 and equation B.9, the quasi-steady equation is then improved by an approximation of this $\chi$ according to,

$$\frac{d\bar{n}}{dt} = -a \frac{\Delta p}{L} + 2 b \left\{ \tau_{w_q} + \chi \right\}.$$  \hspace{1cm} (B.10)

Here the error $\chi$ represents the difference between the unsteady and the quasi-steady wall-shear stress, as defined in equation B.8,

$$\chi = \tau_w - \tau_{w_q}.$$  \hspace{1cm} (B.11)

To solve equation B.8, the correction $\chi$ needs to be approximated. The following approximation was derived by Meuleman [50],

$$\chi_{app} = C \frac{h^2}{\nu} \frac{d\tau_{w_q}}{d\bar{n}} \frac{d\bar{n}}{dt}.$$  \hspace{1cm} (B.12)

where $C = 1/60$. The exact correction $\chi$ as well as the approximation of the correction $\chi_{app}$ are dependent on the dimensionless Stokes number. Meuleman [50] has shown that the correction $\chi_{app}$ applies for $Stk < 2$, and that it should become smaller for $Stk > 2$ and eventually become zero for large Stokes numbers. To make the approximated correction $\chi_{app}$ become smaller for large Stokes numbers and eventually become zero, a relaxation equation of this term is suggested according to

$$\frac{d\bar{n}}{dt} = -a \frac{\Delta p}{L} + 2 b \left\{ \tau_{w_q} + \chi \right\}$$  \hspace{1cm} (B.13)

$$\tau \frac{d\chi}{dt} = C \frac{h^2}{\nu} \frac{d\tau_{w_q}}{d\bar{n}} \frac{d\bar{n}}{dt} - \chi.$$  \hspace{1cm} (B.14)

The time necessary for the core flow to adapt to the new velocity is defined by the chosen relaxation time scale $\tau$. The relaxation time scale $\tau$ is determined from the Stokes number in equation 5.10,

$$\tau = \frac{h^2 \pi}{2 \nu Stk^2}.$$  \hspace{1cm} (B.14)

Meuleman [50] has shown that this relaxation equation applies very well for time scales between the two extremes, corresponding to the Stokes numbers $2 < Stk < 200$. For large time scales the old solution without correction is obtained, as in equation B.9, and for small time scales the correction has a positive influence on the solution. To be able to evaluate this correction, Meuleman [50] has performed experiments in a water channel facility. According to these experiments $Stk = 20$ appeared to be appropriate value for determination of the relaxation time scale.
Acknowledgments

First of all, I would like to thank my daily supervisor and co-promotor dr. Rick de Lange and my promotor prof. Anton van Steenhoven for giving me the opportunity to do my PhD on one of the research topics in their group, for the excellent supervising, for all the help and support and for the trust they had in me during the four years of the PhD programme.

I would like to thank ir. Jan Smeulers and dr. Rene Peters for their supervising and support from TNO and for helping me to realize my goals within the project. My thanks also go to ir. Frederic Breussin from TNO, who helped me several times to realize the numerical model within FLUENT. I would also like to thank prof. Maarten Steinbuch, dr. Bram de Jager and ir. Jan van Helvoirt from the group Control Systems Technology at the Eindhoven University of Technology for their partnership, support and contribution to this project. Furthermore, I would like to thank all the people from Siemens Demag Delaval Turbomachinery who were involved in this project. I especially would like to thank ir. Eric Borg and dr. Theodor Wallman who took care of all the correspondence and supplied me with necessary information.

I am very thankful to Frank van Boxtel who dedicated his MSc thesis to the visualization of the core-flow instability. I would like to thank him for his effort and hard work in order to realize the measurements in the water model of the vaneless diffuser. I would also like to thank the other two students who worked on this research project during their internship, Niels de Koning and Toon Jacobs.

My thanks also go to all the colleagues from Laboratoire de Mécanique de Lille in France. I would like to thank prof. Gerard Bois from ENSAM for the initiation of the cooperation between Laboratoire de Mécanique de Lille and Eindhoven University of Technology at the 6th European Conference in Lille in 2005. I would also like to thank prof. Guy Caignaert from ENSAM for his effort to realize the measurements in Lille and for his guidance. Furthermore, I would like to thank prof. Patrick Dupont from Ecole Centrale de Lille, dr. Antoine Dazin from ENSAM and ing. Sébastien Coudert from CNRS for their help and for giving me the opportunity to use their test rig and equipment at Laboratoire de Mécanique de Lille.

I would like to thank prof. Bert Brouwers, prof. Harry Hoeijmakers and prof. Guy Caignaert for thoroughly reading my thesis and for their valuable comments. Furthermore, I would like to thank prof. Yoshinobu Tsujimoto for his good will, time and effort helping me to understand his two-dimensional inviscid analysis that I have used for comparison in this thesis.

I would also like to thank my direct colleagues for showing me the way and giving me advise and help that I needed during those four years. Thanks to my fellow PhD students for sharing and exchanging their experiences and impressions with me. Furthermore, I would like to thank my friends who showed understanding and gave me support, especially during the last year of the PhD.

I would like to give the greatest thanks to my mother Mira Ljevar and my father Anton Ljevar for giving me support in all possible ways. Without their love, encouragement and support, I would not have come as far as I did today. They have proven themselves to be my true guardians.
and lifetime teachers. Special thanks go to my elder brother Boris Ljevar, who taught me to be a tough sister and who I always have looked up to.

Finally, a very special thanks go to my sweet boyfriend Carlo Schat, for his love, encouragement and support, but also for his sacrifice, which was necessary to complete this PhD thesis in time. Knowing how critical I am about my own work, Carlo has taken the responsibility to make the cover of this thesis. For this I thank him very much, since it gave me more time to work on the contents of this PhD thesis.

I have very much enjoyed most of the PhD programme at the Eindhoven University of Technology, and I will look back with pleasure at those years. Thanks to all the people who worked with me during that time, and thanks to all the people and friends who supported me during this period of my life.

Svetlana Ljevar

January 2007
Eindhoven, the Netherlands
Résumé

Svetlana Ljevar was born in Banja Luka, former Yugoslavia, on 19th of February 1978. From 1991 to 1996 she followed secondary education, of which one year at Gymnasium in Kladovo (former Yugoslavia), one year at Jan van Berlaer MAVO in Helmond (the Netherlands) and the rest of the time at Jan van Brabant HAVO in Helmond.

From 1996 to 2003 she studied Mechanical Engineering, of which one year at the Higher Technical School Midden-Brabant in Tilburg, and the remaining time at Eindhoven University of Technology. She specialized in the field of fluid mechanics. As a part of her study, she did a bachelor training at the Eindhoven University of Technology and an internship at Level Energietechniek B.V. in Son, the Netherlands. The bachelor training concerned a study on optical measurement methods of temperature in glass during glass production, where the influence of the heat loss through radiation and conduction on the temperature gradient in the cylindrical specimen was studied. The internship concerned the development of the Vacuum Isolation Panels, where analysis on the thermal deformation of the panels and occurring tensions due to the temperature difference across the panels is performed.

Svetlana performed her graduation project in the section of Process Technology at the department of Mechanical Engineering at Eindhoven University of Technology. The graduation project involved performance analysis of small diameter turbine mass-flow meters, the so-called IR-Opflow sensors, in order to investigate their non-linear behavior and relatively large inaccuracies within their working range.

In January 2003, Svetlana started her PhD project in the section of Energy Technology at the department of Mechanical Engineering at Eindhoven University of Technology. The PhD project is supported by TNO Science and Industry (Nederlandse Organisatie voor toegewatte-natuurwetenschappelijk onderzoek) and concerns the modeling of rotating stall in wide vaneless diffusers of centrifugal compressors. This PhD thesis is a result of the project.