Rotating stall in a two-dimensional vaneless diffuser flow

Citation for published version (APA):

Document status and date:
Published: 01/01/2005

Document Version:
Accepted manuscript including changes made at the peer-review stage

Please check the document version of this publication:

• A submitted manuscript is the 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

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.
rotating stall in a two-dimensional vaneless diffuser flow

S. Ljevar - H.C. de Lange - A.A. van Steenhoven

Energy Technology Section, Department of Mechanical Engineering
Eindhoven University of Technology, Eindhoven, the Netherlands
s.ljevar@tue.nl, h.c.d.lange@tue.nl, a.a.v.steenhoven@tue.nl

abstract

This paper reports a numerical investigation aimed at understanding the rotating stall mechanism in radial vaneless diffuser. The study of vaneless diffuser flow instability is restricted to a two-dimensional flow analysis, where the influence of wall boundary layers is neglected. A commercial code with the standard laminar viscous flow solver is applied to model the incompressible vaneless diffuser flow in the plane parallel to the diffuser walls. At diffuser inlet rotating jet-wake velocity profile is prescribed and at diffuser outlet constant pressure. Current study reveals that a two-dimensional rotating instability in the vaneless radial diffuser occurs when the critical flow angle is reached. The diffuser flow stability limit is determined for different diffuser radius ratios, which reveals that the core flow stability in the vaneless radial diffuser improves as the diffuser radius ratio decreases.

nomencalature

Greek letters

$c_p$ specific heat at $p = \text{const.} \ [\text{J kg}^{-1}\text{K}^{-1}]$

$\alpha$ flow angle [$^\circ$]

$\gamma$ ratio of specific heat [-]

$\eta_i$ overall isentropic efficiency [-]

$\nu$ kinematic viscosity [$\text{m}^2\text{s}^{-1}$]

$\theta$ tangential coordinate [-]

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

$\sigma$ slip factor [-]

$\psi$ power input factor [-]

Subscripts

$cr$ critical value

$m$ mean value

$i$ impeller inlet

$t$ diffuser inlet

$2$ diffuser outlet

$3$ diffuser outlet

Introduction

Vaneless diffuser rotating stall is one of the unstable flow phenomena that limit the operating range of centrifugal compressors at low mass flows. To increase the region of operation insight into the flow dynamics of rotating stall in the vaneless diffusers is required.
In the literature different approaches have been used to investigate rotating stall such as the wall boundary layer approach and the two-dimensional diffuser flow approach where the wall boundary layers are excluded from the analysis. Several theories that explain the vaneless diffuser rotating stall mechanism have been presented, but also assumptions are made that two or maybe more different flow mechanisms might exist that can cause the rotating stall instability. Mostly one mechanism is associated with the core flow or the jet-wake instability occurring when the critical flow angle is reached and the other is associated with the three-dimensional wall boundary layer instability. Besides the analytical analyses a lot of experimental techniques are used to measure rotating stall within the vaneless radial diffusers. Experimental work has shown the significant influence of diffuser geometry on the vaneless diffuser performance. In more recent work also numerical studies on rotating stall can be found although results on rotating stall in vaneless diffusers are still rare.

As mentioned many authors have used the three-dimensional boundary layer approach in their analytical analysis. Jansen (1964) found that unsteady flows would occur when a three-dimensional flow separation exists and that the location of flow separation depends on the flow angle, the inlet Reynolds number, the Mach number and the diffuser geometry. Senoo et al. (1977) examined the influence of the diffuser geometry and the inlet flow conditions on the critical flow angle for reverse flow and found that the diffuser width ratio, the inlet Mach number and the distortion of the inlet velocity distribution have significant influence on the critical flow angle, while Reynolds number and the boundary layer thickness at the inlet have minor influence. According to Frigne and van den Braemussche (1985) a transient perturbation of the static pressure distribution at the diffuser outlet will induce the rotating flow oscillation if the absolute inlet flow angle has reached a critical value and the periodicity of the perturbation corresponds to the experimentally observed value. Dou (1998) found for wide vaneless diffusers that when reverse flow zone in the wall boundary layer appears at the rear part of the diffuser no rotating stall is generated and that rotating stall is only generated when the reverse flow extends close to the entry region.

Experimental studies have shown that the diffuser geometry strongly influences the diffuser performance, but it also made some investigators presume that different flow mechanisms might exist which can lead to the occurrence of rotating stall in vaneless diffusers. Dou (1991) and Abdelhamid and Bertrand (1980) both found that the instability behavior of the vaneless diffuser is different for narrow and wide diffusers. According to You Hwan et al. (1998) two different mechanisms exist, one dominated by the extension of the reentering flow from the diffuser exit and the other dominated by the growth of the local flow separation zone on the hub and shroud side.

Different results from the mentioned literature indicate that the rotating stall phenomenon in vaneless diffusers is still obscure and that further research on this topic is needed. Since some literature suggests that the core flow instability might be one of the mechanisms that cause the rotating stall in vaneless diffusers a two-dimensional diffuser flow instability analysis is performed to investigate this. The rotating stall inception is investigated from the point of view that it can be caused by a two-dimensional core flow or jet-wake instability. For comparison, a few analytical analyses are found that also use a two-dimensional diffuser flow approach. These are the work of Abdelhamid (1980) and Tsujimoto et al. (1996) who both used a two-dimensional inviscid and incompressible flow analysis to study the vaneless diffuser rotating stall.

This paper does not discuss any experimental research, but to understand the texture of the numerical model a few words are spent on the relation between the experimental and numerical model. To experimentally study the overall image of rotating stall a water model of the centrifugal compressor stage is build. The hydrodynamic analogy shows that a water model of the same geometry must operate at much lower fluid velocities and shaft speeds than the air configuration. Deceleration of the process allows plain visualization of the unsteady flow phenomena using the particle image velocimetry. Therefore, scaling of the existing air compressor stage into a water model of the centrifugal compressor stage is performed and applied. The operating condition of the
existing research air compressor at the stability limit is scaled to obtain a water model that operates near the stalling conditions. Since the numerical model is destined to support the experimental study of rotating stall it is based on the scaled diffuser geometry and fluid flow conditions.

In this paper a numerical analysis of the two-dimensional vaneless diffuser flow instability is performed using the computational fluid dynamics. Performed analysis is applicable only to wide diffusers where the influence of the wall boundary layers can be neglected. This analysis involves the generation of the rotating stall mechanism and the influence of the diffuser radius ratio on the stability limit of the vaneless diffuser core flow. This paper is divided in several sections. First section describes the applied scaling procedure and is followed by the section about the texture of the numerical model. Model results are discussed in a sequence of sections about the occurred two-dimensional rotating instability, the stability limit, the generation mechanism and the influence of the diffuser radius ratio on the stability of the vaneless diffuser core flow.

SCALING

First, scaling of the working medium is performed. Therefore, the pressure rise across the air compressor, $p_{03}/p_{01}$, and the compressor working with water, $p_{3}/p_{1}$, is set equal assuming that the diffuser dimensions remain unchanged. For the air compressor configuration

$$
\frac{p_{03}}{p_{01}} = \left[ 1 + \frac{\eta_c \cdot (T_{03} - T_{01})}{T_{01}} \right]^{\frac{\gamma}{\gamma-1}} = \left[ 1 + \frac{\eta_c \cdot \psi \cdot c_p \cdot \eta_{ip}^2}{c_p \cdot T_{01}} \right]^{\frac{\gamma}{\gamma-1}},
$$

according to Cohen et al. (1996).

When working medium is water pressure rise across the compressor is defined by the energy equation,

$$
\left( \frac{p_3}{p_1} \right) = 1 + \frac{\rho \cdot c_p \cdot \eta_{ip}^2}{p_1} \cdot \left( V_3^2 - V_1^2 \right).
$$

Operating conditions of the water model are obtained by substitution of the air compressor operating conditions at stall onset.

Subsequently, the dimensions of the water model are scaled to the desired values. Therefore, the pressure gradient across the diffuser is kept constant. The pressure gradient across the diffuser is defined as follows

$$
\frac{p_3 - p_2}{r_3} = -\frac{1}{2} \cdot \frac{\rho}{r_3} \cdot (V_3^2 - V_2^2).
$$

Keeping the pressure gradient constant and applying the diffuser radius ratio $r_1/r_2 = 1.52$ gives the relation between the absolute velocity at the diffuser inlet and the diffuser outlet radius, $V_3^2/r_3 = 121.86$, which must apply to the new diffuser geometry. Furthermore, constant mean flow angle must apply and the similarity of the velocity triangles is applied at the flow condition near the stability limit. The velocity triangle relations are used to determine the water model operating conditions.

To obtain exact similarity between the air compressor flow and the fluid flow within the water model the Reynolds and Mach numbers must remain unchanged. Since the pressure gradient is kept constant in the scaling procedure it is not possible to also keep the Reynolds numbers exactly the
same in the water and air configuration. One can satisfy either the Reynolds number condition or the pressure gradient condition but not both conditions at the same time. Check of the flow field similarity showed that the Reynolds number, \( \text{Re} = r \cdot V / \nu \), in air and water equals the order of \( 10^5 \) or higher. The fluid flow as well as the air compressor flow is of a highly turbulent character. It is not possible to satisfy the Mach number similarity when water is used as working medium. In the air configuration \( Ma < 1 \), which means that the flow is subsonic. When the flow is subsonic, density changes over the compressor stage are small and the gas dynamic phenomena like shock waves do not have to be considered.

**NUMERICAL MODEL**

A two-dimensional fluid flow in the \( r-\theta \) plane of the vaneless diffuser is modeled where the influence of the wall boundary layers is excluded. The modeled vaneless diffuser has the inlet and outlet radius at 0.03225 and 0.04908 [m] respectively. A simple two-dimensional quadrilateral grid consisting of 750 by 62 elements is applied to this geometry. At the diffuser inlet a velocity profile as shown in figure 1 is specified. The tangential velocity component is constant while the radial velocity component is described by the rotating hyperbolic tangent function. The impeller rotation direction is clockwise. At the diffuser outlet a constant static pressure is assumed. At the reference situation operating conditions near the stability limit are applied, which are obtained from the scaling procedure. The reference tangential and corresponding mean radial velocity component are \( \nu = 2.4282 \) and \( \mu = 0.2905 \) [m/s] respectively.

For the numerical analysis a commercial software package, the Fluent6 code from Fluent Inc., was used. The governing integral equations for the conservation of mass and momentum are solved using the finite-volume approach. For the time dependent terms, the second-order implicit unsteady formulation is used. In this situation four discretisation schemes for the convection terms are available to choose from. The solutions of the four discretisation schemes, represented by the velocity vectors, are given in figure 2. The first order upwind and the power law scheme are first order accurate, the second order upwind is second order accurate and the QUICK scheme is third order accurate for the spatial discretisation of the convection terms and second order accurate for the diffusion terms. Figure 2 shows that the solutions obtained with the power law and first order upwind scheme are damped out compared to the solutions obtained by the other two schemes. It is assumed that the damping effect could prevent the occurrence of instability and the higher order discretisation schemes for convection terms are preferred. There are negligibly small differences in solution between the 2nd order upwind and the QUICK scheme. Eventually to avoid excessive numerical dissipation the higher order QUICK scheme is used in the model.

The Reynolds number, \( \text{Re} = r \cdot V / \nu \), is about \( 10^5 \) throughout this study. Although the studied fluid flow is turbulent, the incompressible laminar viscous model is used. Since the turbulence models capture the diffusion-like character of turbulent mixing associated with many small eddy structures it is not known if they will give the same results as the laminar model, but to avoid excessive numerical dissipation laminar viscous model is chosen for this analysis. It is assumed that
higher amount of viscosity will have damping effect on the solution and that it will be difficult for the instability of large structures like this one to occur in turbulence models. It is expected that the use of turbulence models probably would result in delay of instability inception.

![Figure 2: The discretisation schemes for the convection terms](image)

**2D ROTATING INSTABILITY**

Within this two-dimensional diffuser flow analysis the tangential and mean radial velocity component are varied. The analysis has shown that the impeller speed i.e. tangential velocity component has significant influence on the two-dimensional vaneless diffuser flow. Solutions for different impeller speeds, represented by the velocity colored by velocity magnitude, are given in figure 3. For each solution the corresponding mean flow angle $\alpha_m$ at the diffuser inlet is given.

![Figure 3: The influence of the impeller speed](image)
As the impeller speed becomes very small stable steady solution of the diffuser core flow slowly transforms towards the solution of the impeller at rest, as shown in figure 3. At low impeller speeds mean radial velocity becomes proportional to the tangential velocity and the jet-wake pattern can move directly towards the diffuser outlet instead of being smeared out first and then propagate slowly towards the outlet. On the other hand, at high impeller speeds a two-dimensional rotating instability similar to rotating stall occurs. The obtained two-dimensional rotating instability fully develops within four to six impeller revolutions. For the given diffuser radius ratio of \( r_3/r_2 = 1.52 \) and \( N = 17 \) it consists of six rotating cells that propagate with approximately 40 % of the impeller speed in the absolute frame of reference. Because of these characteristics it is presumed that the occurred two-dimensional flow instability might contribute to the diffuser rotating stall.

**STABILITY LIMIT**

In search for the critical flow parameter, a flow parameter that determines the stability limit of the two-dimensional vaneless diffuser flow, the tangential and mean radial velocity component are varied alternately for a given diffuser radius ratio \( r_3/r_2 = 1.52 \). A matrix of simulations is performed showing that a two-dimensional rotating instability, similar to rotating stall, occurs when \( v/u_m \) ratio at the diffuser inlet exceeds a value of approximately 8.4. The \( \Delta p-q \) plot for the performed simulations is given in figure 4 with on the lines of constant impeller speed attached the \( v/u_m \) ratio corresponding to each simulation.

![Figure 4: Stability limit for the diffuser radius ratio \( r_3/r_2 = 1.52 \)](image)

In figure 4 the pressure rise across the diffuser seems to be independent or slightly decrease with decreasing flow rate. This is not expected for the radial diffuser. It can be explained due to the fact that no effects of the wall boundary layers are taken into account in the model. In the three-dimensional case the shear layers at the walls create a resistance that has to be overcome. At lower
mass flows the resistance as well as the pressure losses decrease, which results in higher pressure rise across the diffuser at low mass flows than at high mass flows.

Since the \( \nu/u \) ratio can be interpreted as the flow angle \( \alpha \), as defined in figure 1, the stability limit of the two-dimensional vaneless diffuser flow is expressed in terms of the critical flow angle at diffuser inlet, \( \alpha_{cr} \). The core flow stability criterion suggests that the two-dimensional rotating instability will occur as soon as the mean flow angle at the diffuser inlet becomes smaller than the critical flow angle: \( \alpha_m < \alpha_{cr} \). For the wide vaneless diffusers with \( r_3/r_2 = 1.52 \) and with current approximation of the diffuser inlet- and outlet flow conditions it is found that \( \alpha_{cr} = 6.8^\circ \).

**TRANSITION MECHANISM**

Transition from the stable steady diffuser flow into the two-dimensional rotating instability is shown in figure 5 for the constant impeller speed of 495 [rpm]. The mean radial velocity component decreases with continuous linear function from 0.1714 to 0.1309 [m/s] within a time period of 1 [s]. The figures show the velocity field colored by velocity magnitude. As the mean radial velocity component i.e. mean flow angle at the diffuser inlet is being decreased the stable rotating flow pattern becomes unstable. Its structure falls apart and after few impeller revolutions a new on itself stable flow condition is observed. For each image in figure 5 the passed time \( t \), the mean flow angle at diffuser inlet \( \alpha_m \) and the value of the mean radial velocity \( u_m \) at that point are given.

![Figure 5: Diffuser flow for decreasing mass flow at constant impeller speed](image)

To give an impression of the obtained vaneless diffuser flow field a close-up of the diffuser flow is given in figure 6 for the stable and unstable situation corresponding to the solutions 1 and 6 from figure 5. At stable operating condition a stronger outward pointed and a weaker reversed flow area seem to alternate near the diffuser outlet. The number of these regions exactly equals the number of the impeller blades. When \( \alpha_m < \alpha_{cr} \) the diffuser flow is strongly circumferential and some of the
fluid flow is able to pass under the alternating pattern and causes the alternating flow areas to become unequal in size. When $\alpha_m$ is further decreased the pattern will fall apart as shown in figure 5. From this point smaller and weaker high velocity regions seem to merge with the larger high velocity regions, which makes these larger regions grow bigger. Finally, a number of these rotating cells will reach their final size and rotational speed and distribute equally around the circumference to form another stable operating condition. This way of transition from the steady stable solution to the two-dimensional rotating flow instability indicates that the instability originates near the diffuser outlet and is dependent on the diffuser outlet flow conditions.

DIFFUSER RADIUS RATIO
By changing the diffuser outlet radius the diffuser radius ratio is varied to investigate its influence on the critical flow angle. The obtained critical values for different radius ratios are given in figure 7 where they are compared to the values obtained by Tsujimoto et al. (1996) for $m = 1, 2$ and $3$. Tsujimoto et al. (1996) who also performed a two-dimensional vaneless diffuser flow analysis found that vaneless diffuser flows have a two-dimensional, inviscid and rotational flow instability and that the critical flow angle is a function of the diffuser radius ratio and number of stall cells. The major difference between the current analysis and that of Tsujimoto et al. (1996) is that he generated the two-dimensional rotating instability by prescribing the uniform velocity profile at the diffuser inlet and adding a one-, two- or three-cell disturbance to it while the current analysis prescribes the jet-wake velocity profile at the diffuser inlet which becomes unstable when the critical flow conditions are reached. Both analyses prescribe constant pressure at the diffuser outlet. Despite the differences in approach both studies reveal the occurrence of the two-dimensional rotating flow instability in the region where the mean flow angle $\alpha_m$ is smaller than the critical flow angle $\alpha_{cr}$ and the obtained critical flow angle values are similar.
for \( r_3/r_2 \leq 2 \). The differences in obtained relations in figure 7 can not be exactly explained but it is suspected that the different diffuser inlet boundary conditions are the cause of that.

The current study shows that the critical flow angle increases with increasing diffuser radius ratio. The core flow stability improves as the diffuser radius ratio decreases. For larger diffuser radius ratios, \( r_3/r_2 \geq 2.5 \), the critical flow angle is large and the diffuser flow is found to be unstable at all operating conditions until the impeller speed becomes so low that \( v/u_m < 2.0 \), which gives the solution of the nearly standing still impeller as shown in figure 3.

As the diffuser radius ratio and subsequently the distance between diffuser inlet and alternating flow pattern near diffuser outlet increases the critical flow angle also increases. This again indicates the significant influence of the diffuser outlet flow conditions on the generation of a two-dimensional rotating instability.

CONCLUSIONS

In this paper a numerical analysis of the two-dimensional instability of the vaneless radial diffuser is performed. The analysis is performed to investigate if the core flow instability can be the cause of rotating stall in vaneless radial diffuser of the centrifugal compressor. A two-dimensional rotating instability associated with rotating stall in radial vaneless diffusers is found to exist. The results show that the stability limit for wide vaneless diffusers, where the effect of the wall boundary layers is negligible, can be expressed in terms of the critical flow angle at the diffuser inlet, \( \alpha_{cr} = \tan^{-1}(u_m/v)_r \). When the mean flow angle at the diffuser inlet becomes smaller than \( \alpha_{cr} \) a two-dimensional rotating instability occurs. The results show that the obtained stability limit is influenced by the diffuser radius ratio. As the critical flow angle decreases with decreasing diffuser radius ratio the stability of the vaneless diffuser core flow improves. This is in good agreement with Abdelhamid and Bertrand (1980) who found that the stability limit is strongly dependent on the diffuser radius ratio. Good agreement is also obtained with Tsujimoto et al. (1996) in the sense that both analyses found that a two-dimensional rotating instability occurs when the mean flow angle becomes lower than the critical flow angle and that the critical flow angle increases with increasing diffuser radius ratio. Numerical analysis also reveals that a two-dimensional rotating instability strongly depends on the diffuser outlet flow conditions. The point of inception depends on the diffuser radius ratio i.e. the diffuser outlet flow conditions, which can agree with the work of You Hwan et al. (1998) who found that reentering flow from the diffuser exit might be one of the flow mechanisms responsible for the rotating stall inception in vaneless diffusers.

Since the vaneless diffuser flow is three-dimensional the applied two-dimensional approach has its limitations. Any three-dimensional flow mechanisms that might be responsible for the occurrence of rotating stall can not be detected with this model. Therefore the model must be extended with the wall boundary layers at the hub and the shroud side wall. It is expected that addition of the wall boundary layer effects into the model will probably change if not the magnitude of the critical flow angle then certainly the structure of the two-dimensional rotating instability, i.e. the number and propagation speed of rotating cells. It is not known if the three-dimensional effects are more determinative for the stability limit or the structure of rotating instability or for both, but it is to be investigated.

This analysis does not completely explain the mechanism of rotating stall, but it suggests a few of the physical phenomena and mechanisms of rotating stall. The overall impression of rotating stall mechanism gained from the performed analysis and the results found in the literature is that more different flow mechanisms might exist that could lead to the occurrence of rotating stall in vaneless radial diffusers. The scenario of two different mechanisms is very possible, but must be further investigated. One mechanism is associated with the core flow or the jet-wake instability occurring in wide diffusers when the critical flow angle is reached and the other is associated with the three-dimensional wall boundary layer instability occurring in the narrow diffusers. The work of
Abdelhamid and Bertrand (1980), You Hwan et al. (1998) and Dou (1991) points towards the similar suggestions.

To clarify the flow pattern of rotating stall in vaneless diffusers also the three-dimensional flow analysis must be performed besides this two-dimensional analysis. First, to fully understand the contribution of the two-dimensional effects, a two-dimensional analysis will be completed. The influence of the inlet flow conditions such as the Reynolds number, the intensity and shape of the jet-wake pattern, the number of impeller blades and the velocity fluctuations on the critical flow angle still must be investigated as well as the influence of the diffuser geometry and the mentioned inlet flow conditions on the two-dimensional rotating instability i.e. the number, speed and extent of propagating cells. Then the three-dimensional flow analysis including the wall boundary layer effects will be performed.

This study on rotating stall mechanism will be continued using not only the numerical but also the experimental techniques. A water model of the centrifugal compressor stage will be made in order to perform rotating stall measurements suitable for comparison with the numerical analysis.

ACKNOWLEDGMENTS

TNO TPD from Delft is thanked for their contribution to this project as well as Siemens Demag Delavst Turbomachinery B.V. from Hengelo for making their compressor data and the research air compressor available.

REFERENCES

Abdelhamid, A.N.; (1980); Analysis of Rotating Stall in Vaneless Diffusers of Centrifugal Compressors; ASME Paper 80-GT-184; New Orleans.

Abdelhamid, A.N. and Bertrand, J.; (1980); Distinctions Between Two Types of Self-Excited Gas Oscillations in Vaneless Radial Diffusers; ASME Paper No. 79-GT-58; Canadian Aeronautics and Space Journal; volume 26, pp. 105-117.


Jansen, W.; (1964); Rotating Stall in a Radial Vaneless Diffuser; Journal of Basic Engineering, Transactions of the ASME; volume 86, pp.750-758.


You Hwan Shin; Kwang Ho Kim and Byung Jin Son; (1998); An Experimental Study on the Development of a Reverse Flow Zone in a Vaneless Diffuser; JSME International Journal, Series B; volume 41, pp. 546-555.