Evolution of unsteady vortex structures and rotating stall cells in a centrifugal compressor with vaneless diffuser

Siya Jiang
Tsinghua University
jiang-sy12@tsinghua.org.cn
Beijing, China

Cheng Tian
Tsinghua University
tianc19@mails.tsinghua.edu.cn
Beijing, China

Song Fu
Tsinghua University
fs-dem@tsinghua.edu.cn
Beijing, China

ABSTRACT
The rotating stall in a centrifugal compressor with a vaneless diffuser is investigated using the IDDES (Improved Delayed Detached Eddy Simulation) method. The flow fields from design to off-design conditions are assessed for comparison. The characteristic frequency of the rotating stall is accurately predicted, illustrated by both the pressure spectra and the velocity signals. This low-frequency peak corresponds to the stall cells, induced by different vortex structures in the impeller passages and the diffuser. Compared with the flow fields of the rotating instability and the near design conditions, a unique flow structure of the rotating stall condition is the longitudinal vortices upstream of the impeller inlet, connecting to the tornado-type vortices in one or two consecutive passages. In the downstream passages, the large-scale vortices on the shroud side cause the blockage. The deformed vortices stretching along streamlines towards the outlet are dominant in the diffuser. Through the high fidelity numerical method, physical mechanisms are analyzed in detail to give a new perspective of the rotating stall in a centrifugal compressor.

INTRODUCTION
When centrifugal compressors operate under restricted conditions, such as high pressure ratios and low flow rates, the unstable flow fields through blades result in harmful consequences including efficiency loss and even destructive vibrations (Marshall and Sorokes, 2000). The occurrence of rotating instability and rotating stall phenomena is a crucial concern for under-performing situations. In this sense, the investigations for the rotating stall flow mechanisms are of great importance to the design process.

Researchers’ focuses have been concentrated on the rotating stall phenomenon in the axial compressors for decays and numerous meaningful agreements have been reached. The rotating stall was proved to be relevant to the breakdown of tip leakage vortex (Yamada et al., 2004). Separation regions, which are always found near the leading edge on the suction side, interact with the tip leakage vortex. The interaction results in a tornado-type vortex with a radial vortex leg attached to the casing and vortex head to the suction side of the blade (Yamada, 2012). The vortex leg propagates toward the pressure side of the neighbouring blade and induces a new vortex (Inoue et al, 2001).

For the centrifugal compressors, the performance undergoes a mild change as the rotating stall takes place rather than the abrupt decreasing of pressure ratio in axial compressors. This is because the centrifugal force of the impeller can overcome the adverse pressure gradient and maintain the stability of the compressor. As a consequence, the rotating stall develops from a more complicated situation with flow recirculation in the impeller (Harley et al., 2015). The intense secondary flow in the impeller also affects the rotating stall in the diffuser by causing circumferential distortion of its inlet flow, resulting in the hub-corner separation on diffuser vanes (Marsan et al., 2015). The circumferential non-uniformity for most centrifugal compressors is caused by volute governs the evolution of the rotating stall cells (Niu et al., 2020; Yang...
et al., 2019). The generation and disappearance process of rotating stall cells are observed due to the pressure distortion along the circumferential direction in a vaneless diffuser.

Except for RANS (Reynolds Averaged Navier-Stokes) and experimental methods, high fidelity numerical methods were proved to be the advanced alternatives for the investigations of the flow structures in rotating stall conditions (Zhou, 2017). The differences with axial compressors in the detailed flow structures concerning the rotating stall phenomena are investigated. The tornado-type vortices, illustrated by Bousquet et al. (2016) using LES (Large Eddy Simulation), are associated with the mode-type stall inceptions rather than spike-type inceptions, which is inconsistent with the previous theory for axial compressors. Therefore, the rotating stall in centrifugal compressors manifests some different features from the axial compressor. Whether the conclusions in the flow mechanisms for axial compressors can be extended is doubted and needs more investigation.

In order to investigate the flow mechanisms, Fujisawa et al. (2018) conducted a numerical simulation using the DES-type method. A tornado-type vortex was observed at the impeller inlet related to a stage stall cell. Blockages were found on the hub side at the throat of impeller passages and in the vaneless diffuser. However, Iwakiri et al. (2009) attributed the blockage of the passages to the breakdown of the full-blade tip leakage vortex at developed stall conditions. For the developed rotating stall conditions close to the surge line, the flow separation went in the upstream direction, inducing a strong shear-layer interface and seeded streamlines on the inner side of the interface. The strength of the generated shear layer in the shroud entrance was decisive for the rotating stall (Sundstrom et al., 2018).

In conclusion, the current studies are far from sufficient for the understanding of the flow mechanisms for centrifugal compressors. Although the high fidelity methods are expensive for a full-annular simulation, they have made great contributions to the revelation of the physical sense. The present paper investigates the rotating stall phenomenon for a centrifugal compressor using the IDDES method. It is aimed at investigating the flow mechanisms of the rotating stall in a centrifugal compressor and making a supplement for the current researches.

**METHODOLOGY**

The current case is based on the experiment of an in-house designed centrifugal compressor, consisting of an impeller, a vaneless diffuser and a volute as described in detail by Niu et al. (2020). The bolt and nut at the inlet hub are also considered as illustrated in Figure 1(a). Table 1 lists the specifications of the compressor.

<table>
<thead>
<tr>
<th>Table 1 Dimensions of the centrifugal compressor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotational speed</td>
</tr>
<tr>
<td>Number of blades</td>
</tr>
<tr>
<td>Impeller leading edge tip radius</td>
</tr>
<tr>
<td>Impeller leading edge hub radius</td>
</tr>
</tbody>
</table>
Both RANS and IDDES are carried out for different operating conditions. Steady RANS is used for case 1 at the designing points and case 2 at the near-stall point, marked in Figure 2(a). The RANS result is also used as the initial condition for case 4 with IDDES at the rotating stall condition. The detailed numerical methods for both of them are briefly described respectively.

**Numerical methods for IDDES**

An in-house code is used for IDDES based on the finite volume method in a rotating Cartesian frame fixed on the rotor. The third-order MUSCL (Monotonic Upwind Scheme for Conservation Laws) interpolation by Van Leer (1985) is used to discretize the convective terms in momentum equations and the centred scheme is employed for diffusion terms. Implicit LU-SGS with sub-iterations in pseudo time (rTS-LU-SGS) is chosen as the temporal marching method. IDDES method proposed by Shur et al. (2008) is based on the SST k-ω model to avoid the grid-induced separation. The computational domain contains all 24 blades. A full annular analysis is carried out and circumferential non-uniformity is considered. The inlet boundary is located 115.63mm upstream of the leading edge. The total pressure of 101325Pa and the total temperature of 288.15K is defined as the inlet boundary condition. The inlet velocity is along the axial direction (x direction). The outlet boundary is set at the outlet of the diffuser. The volute is replaced by a specific circumferential static pressure distribution for the outlet boundary condition obtained from the hub wall at the diffuser exit, as shown in Figure 2(b). In the direction from the hub to the casing, the static pressure is constant.

The grid from IDDES is exhibited in Figure 1(a). The refinement of the grid is conducted in the tip clearance and the trailing edge. The amount of grid points is 82 million. The first wall-normal grid space $y^+$ is less than 1 in wall units as shown in Figure 1(b). For most of the near-wall grids, the resolutions in the streamwise and spanwise directions are less than 60. The 4 layers of grids near the outlet boundary are used as the damping layer to suppress the nonphysical oscillation from the outlet boundary. No-slip and adiabatic conditions are applied for all the solid walls.

Figure 1(c) shows the instantaneous contour for the blending function $f_j$ for IDDES. The detailed definition is discussed in Shur et al.’s (2008) paper. When the blending function equals 1, the IDDES hybrid method is effectively equivalent to the RANS branch. Otherwise, the LES branch comes into effect. The figure indicates that the attached boundary layer is adopted as the RANS region, and the main flow and the separated flow is adopted as the LES region properly.

For IDDES, about 10 thousand time steps are calculated to obtain a convergent solution. $4\times10^5$ more time steps are used for the long-term values of the turbulence statistics. The time step size is 60 time steps for passing one blade passage of impeller.

**Numerical methods for RANS**

The computational domain and the boundary conditions for RANS are the same as IDDES except for two points. Firstly, only one blade passage is considered in the computational domain. Secondly, the circumferential non-uniformity is ignored and constant static pressure is set for outlet boundary condition. As proved by the previous experiment of the similar centrifugal compressor by Iwakiri et al. (2009), the static pressure at the inlet of the volute is almost uniform circumferentially at the near design condition.

SST k-ω model (Menter et al., 2006) is chosen for steady RANS simulation. The convective terms for the $k$ and $\omega$ equations are discretized with the second-order upwind scheme and the diffusion terms are approximated by central differences. The first wall-normal grid space $y^+$ is less than 2. The resolutions in the streamwise and spanwise directions are less than 200. 0.17 million grid points are employed for one passage.
RESULTS AND DISCUSSION

Compressor performance map and fluctuation statistics

The compressor performances obtained by the RANS method and IDDES method are depicted in Figure 2(a), in comparison with the experimental data. The IDDES for rotating stall phenomenon is the focus of the current paper and others are analysed for comparison. The orange triangles are the results for a series of steady RANS, which manifest surprisingly good agreement with the experimental data at the near design conditions. A slight deviation is observed at the choke condition and the near stall condition. The differences from the experimental data become more obvious in the unstable region. The RANS calculations get divergent at about \( m = 0.18 \text{kg/s} \).

The IDDES’s results agree better with the experimental data. For case 4 at the rotating stall condition, the error of pressure ratio is about 3% between the numerical and experimental data. It verifies the reliability of the IDDES results. Case 3 is to predict the rotating instability condition. It also shows a high level of consistency with the experiment.

![Figure 2](image)

**Figure 2** (a) Compressor performance. (b) Circumferential pressure distributions at diffuser exit. (c) Locations of the monitors.

Comparing the performances between case 3 and case 4, a violent oscillation of the performance for case 4 is observed. This case falls in the range of the mass flow rate for the rotating stall in the experiment. The mass flow rate of case 3 is in the range of the stable operating conditions. However, strong fluctuations are also observed as shown in Figure 3(b), to be discussed in detail later.

The monitors for Figure 3 are stationary, being set at the different streamwise positions but the same circumferential position of \( \theta = 313\text{deg} \). At the impeller inlet (marked with “I”), the monitor is set near the shroud, 8mm upstream of the leading edge. At the diffuser inlet (marked with “II”), the monitor is just at the diffuser inlet with \( r = r_1 = 52.50\text{mm} \) at 50% span. The monitor marked with “III” is in the diffuser with \( r = 69.25\text{mm} \) at 50% span. The locations of these stationary monitors are shown in Figure 2(c) marked with yellow stars.

The low-frequency peak \( f_1 \) of 0.43 IPF (Impeller Passing Frequency) in Figure 3(a) confirms to the frequency of the rotating stall in the experiment, which is reported as 0.44 IPF. The peak exits in the impeller passage as well as the diffuser. Near the impeller inlet, its intensity reaches the maximum, and then it goes through a shape decrease in the diffuser.
The high-frequency peak of BPF (Blade Passing Frequency) is observed in the diffuser (agrees with the experiment), while not at the impeller inlet. It may be because of the non-ignorable distance from the monitor to the leading edge. Comparing with the experiment, the most obvious difference lies in the medium-frequency range, where the energy distributes in a broad range between IPF and BPF. This difference can also be reflected by Figure 8 in Niu et al. (2020). At monitor III, the peak of \( f_i = 5.7 \) IPF, which is also found at monitor II, replaces \( f_s \) and becomes dominant.

In Figure 3(b), the low-frequency peak disappears in case 3. Nevertheless, the energy on the broad range of medium-frequency is still at a high level, corresponding to the feature of the rotating instability phenomenon. The energy concentrates on the frequency range of 2 IPF to 7 IPF as emphasized by the dashed box in the figure. The broadband high-frequency noise is a typical indicator of rotating instability as concluded by the previous papers (Mailach et al., 2001, He and Zheng, 2018). Therefore, although the performance of case 3 is close to the experimental results at stable operating condition, as shown in Figure 2(a), we still consider it as the rotating instability conditions.

**Evolution and unsteady behavior of characteristic vortex structures**

In order to investigate the unsteady behaviours in the blade passages, the monitors in this section rotate at the same speed as the impeller. These monitors are arranged in every two passages (in Figure 2(c)), marked with “II-1”~“II-12” at \( r = 73\% r_i \) at the impeller inlet and “II-1”~“II-12” at 89% blade height at the diffuser inlet. In Figure 4(a), apart from the continuous signals with an absolute rotational speed of zero caused by the circumferential non-uniformity, the low rotational speed signals are also seen. The other kind of signals propagates at a speed of 0.43 rotational speed in absolute coordinate, corresponding to a speed of -0.57 of the impeller speed in a rotating system as detected in the figure, consistent with the peak in Figure 3(a). They correspond to the strong reverse flow, indicating the circumferential-propagating rotating stall cells. However, the stall cells can just be detected by the monitors for a short period, implying that the stall cells may undergo a dissipation or a motion departing from the circumferential direction. In Figure 4(b), a similar phenomenon is found at the diffuser inlet, where the stationary continuous signal and the discontinuous signal related to the stall cells are both observed. At the diffuser inlet, the amplitude of the velocity fluctuations increases. Unlike the situation of the impeller inlet, a large number of small-amplitude wave packets mix into the dominant waves causing by the rotating stall cells, suggesting more small-scale flow structures in the diffuser.

The \( \zeta_2 \) criterion (Jeong and Hussain, 1995) is employed to detect three-dimensional vortices, as shown in Figure 5 to illustrate the evolution of the flow structures corresponding to the above-mentioned signals. Because of the non-axisymmetric pressure field caused by the volute, distorted circumferential vortex structures are observed. They are tornado-type vortices, with long vortex legs stretching along streamlines towards the outlet. The stall cells in the vaneless diffuser, which are the low-speed flow regions induced by the dominant vortices, experience a radial propagation as they propagate in the circumferential direction with a gradually decreasing rotating speed towards the outlet.
The impeller inlet is dominant by the large-scale separation regions. The comparisons with the flow structures at the inlet of the impeller in the near design condition (case 1 in Figure 2(a)), near stall condition (case 2), and the rotating stall condition (case 3) are exhibited in Figure 6. In case 1, the long leading edge tip leakage vortices are observed in the main flow direction. These mild vortices do not cause any obvious blockage for the main flow. In case 2, the tip leakage vortices become much stronger since the streamlines show a more prominent spiral feature. The most distinguished difference from case 1 is the occurrence of the separation regions. The strong back-flow vortices are perpendicular to the blade surface close to the pressure side. The vortices interact with the tip leakage vortices, which are inclined toward the circumferential direction. The appearance of the back-flow vortices does not identify with the tip leakage vortex breakdown found in the axial compressor (Yamada, 2004) since the vortex axes are almost perpendicular with the tip leakage vortex.

Under the condition of rotating instability, the interaction between the tip leakage vortices and the separation vortices is so strong that they can hardly be distinguished clearly. Via full-annular IDDES, the unsteady behaviour of the separation vortices is captured. These vortices are observed close to the inlet of the passage near the casing, propagating along the circumferential direction. Due to the relative motion of the separation vortices and the blade surface, these vortices traverse through the tip clearance. The rotational speeds of the separation vortices are not invariable as can be seen from the instantaneous flow field. This corresponds to a broad-range of frequency between 2 IPF to 7 IPF from Figure 3(b).
The flow field in the region is diminished. The tip leakage vortex C becomes non-vortex on the suction side is maintained. At the original long vortex strip of vortex B disappears. The vortex B is thus uncollected with vortex A. Nevertheless, the vortex A further moves, the vortex B' emerges. The original vortex C is weakened as the induced vortex B' alter the inlet flow pattern. The inlet of this passage is thus blocked by a strong separation related to vortex B'. At t = t₀ + 3Tᵢ and t = t₀ + 5Tᵢ, as vortex A further moves, the vortex B' strengthens and vortex C almost disappears. At t = t₀ + 7Tᵢ, the original long vortex strip of vortex B disappears. The vortex B is thus uncollected with vortex A. Nevertheless, the vortex on the suction side is maintained. At t = t₀ + 9Tᵢ, vortex B and B' on the suction further deform, with the separation region is diminished. The tip leakage vortex C becomes non-ignorable or even dominant again. As vortex A corresponding to the stall cell leaves these passages, the inlet flow field generally recovers to the weakly blocked situation similar to the flow field in Figure 6(c).

Figure 6 Comparison of the flow field at the impeller inlet in different operating conditions. (a) to (c) shows the iso-surface of $\lambda_2$ together with streamlines, coloured by relative axial velocity; (a) case 1; (b) case 2; (c) case 3; (d) shows the inlet axial velocity at 8mm upstream of the leading edge.
Figure 7 The vortex structures shown by the contour of $Q$ at 90% span and the iso-surface of $\lambda_2$ at the impeller inlet together with the streamlines in the plane, coloured by relative axial velocity. (a) $t = t_0$; (b) $t = t_0 + T_B$; (c) $t = t_0 + 3T_B$; (d) $t = t_0 + 5T_B$; (e) $t = t_0 + 7T_B$; (f) $t = t_0 + 9T_B$. ($T_B = 1/BPF$)

These vortices induced by the large-scale stall cell not only affect the surrounding flow field at the impeller inlet but also exert further influence on the whole passage. Figure 9 shows the contour of the streamwise velocity at $t = t_0 + 3T_B$. The passage labelled with “(1)” is the passage downstream of vortex B. The contours show the reverse flow region at 50% span. A severe separation is found near the trailing edge of the blades as well as in the diffuser, giving rise to the main flow blockage. The reverse flow region in the passage “(1)” is larger than the other passages related to a throat blockage on the shroud side at the throat of impeller passages. This reverse flow region originates from the casing and blade corner just downstream of vortex B. This proves that the inlet stall cell influences the whole passage by deteriorating the inlet flow field near the casing and thus changing the local angle of attack. The vortex structures in this passage is in Figure 9(a)-(c) in a series of time. The reverse flow region is related to the separation vortices shown in these figures. The separation vortices in the passage are getting weaker when the stall cell moves away and the middle part of passage “(1)” becomes less blocked at $t = t_0 + 7T_B$. In comparison with the blocked the passage “(1)” at $t = t_0 + 3T_B$, the passage “(2)” in Figure 9(d) is unaffected by the stall cells. Therefore, fewer vortices are observed in the passage “(2)” and the main flow attaches to the blade surface. From the discussions above, the stall cell at the impeller inlet blocks the downstream passage and spread from passage to passage in the circumferential direction.
Figure 8 The contours of pressure at 90% span and streamwise velocity at 50% span at $t = t_0 + 3T_B$.

From the discussions above, we find some different features of the vortex structures from the previous researchers. Firstly, the detached boundary layer upstream of the impeller inlet is found and a longitude upstream vortex is the dominant flow structure at the inlet. These vortex structures, as well as the triggered tornado-type vortices on the suction side, are different from the characteristic tornado-type vortices with radial vortex legs in the previous papers. Secondly, the throat blockage is developed on the shroud side. It may be directly caused by the upstream deteriorative inlet flow field near the shroud. This observation does not conform to the flow mechanisms by Fujisawa et al. (2018).

Figure 9 The iso-surface of $\lambda_2 = -2000$ in different passages and time, together with the streamlines, coloured by relative axial velocity. (a). $t = t_0 + 3T_B$, for all ; (b). $t = t_0 + 3T_B$, passage 1; (c). $t = t_0 + 5T_B$, passage 1; (d). $t = t_0 + 3T_B$, passage 2.

CONCLUSIONS

The current paper focuses on the flow mechanisms for the rotating stall phenomenon in a centrifugal compressor using the IDDES method. In comparison with the flow fields in the other operating conditions, the unique features and flow mechanisms possessed by rotating stall are figured out. The conclusions are summarized as follows.

(1). The IDDES results show good agreements with the experimental results on the compressor performance and the unsteady flow features of the rotating stall. The IDDES method is proved to be of higher-level of accuracy for calculating the flow fields in unstable operating conditions.

(2). The vortex structures at the impeller inlet are investigated for different operating conditions. As the mass flow rate decreases, the tip leakage vortices are inclined toward the circumferential direction and separation vortices appear,
interacting with the tip leakage vortices. The propagation of the separation vortices along the circumferential direction corresponds to the broad range of frequency for the rotating instability case.

(3) For rotating stall, a longitudinal vortex is observed upstream of the impeller inlet, inducing tornado-type vortices in one or two consecutive passages. As the longitudinal vortex propagates, the stall cells caused by the typical vortices spread in the circumferential direction. The tip leakage vortices are weakened by the stall cells.

(4) The throat blockage is developed downstream of the tornado-type vortices on the shroud side, caused by the deteriorative inlet flow field near the shroud. The current work describes the rotating stall phenomenon in one typical centrifugal compressor. It can provide one possible flow mechanism for the rotating stall. The different flow features in different numerical cases are to be compared in detail to obtain a universal physical mechanism or criteria for the rotating stall.

**NOMENCLATURE**

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DES</td>
<td>Detached Eddy Simulation</td>
</tr>
<tr>
<td>IDDES</td>
<td>Improved Delayed Detached Eddy Simulation</td>
</tr>
<tr>
<td>RANS</td>
<td>Reynolds-Averaged Navier-Stokes</td>
</tr>
<tr>
<td>LES</td>
<td>Large Eddy Simulation</td>
</tr>
<tr>
<td>$\tilde{f}_d$</td>
<td>blending function for IDDES</td>
</tr>
<tr>
<td>$k$</td>
<td>turbulence kinetic energy [m²/s²]</td>
</tr>
<tr>
<td>$\omega$</td>
<td>specific dissipation rate [s⁻¹]</td>
</tr>
<tr>
<td>$x$</td>
<td>coordinate in the axial direction</td>
</tr>
<tr>
<td>$y^*$</td>
<td>normalized first wall-normal grid space</td>
</tr>
<tr>
<td>$r$</td>
<td>radius [m]</td>
</tr>
<tr>
<td>IPF</td>
<td>Impeller Passing Frequency</td>
</tr>
<tr>
<td>BPF</td>
<td>Blade Passing Frequency</td>
</tr>
<tr>
<td>$f_r$</td>
<td>characteristic frequency for rotating stall</td>
</tr>
<tr>
<td>$T_B$</td>
<td>blade passing period, $T_B = 1/BPF$</td>
</tr>
<tr>
<td>$T_o$</td>
<td>impeller passing period, $T_o = 1/IPF$</td>
</tr>
</tbody>
</table>

**ACKNOWLEDGMENTS**

This work was supported by the National Science and Technology Major Project (Grant No. 2017-II-0004-0016) and the National Natural Science Foundation of China (Grant No. 92052103).

**References**


