Numerical analysis of flow in a centrifugal compressor with circumferential grooves: influence of groove location and number on flow instability

X Chen¹, G Qin¹, Z Ai², Y Ji¹
¹ School of Energy and Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China
² School of Mechanical and Electronic Engineering, Sanming University, Sanming 365004, China
E-mail: glqin@mail.xjtu.edu.cn

Abstract. As an effective and economic method for flow range enhancement, circumferential groove casing treatment (CGCT) is widely used to increase the stall margin of compressors. Different from traditional grooved casing treatments, in which the grooves are always located over the rotor in both axial and radial compressors, one or several circumferential grooves are located along the shroud side of the diffuser passage in this paper. Numerical investigations were conducted to predict the performance of a low flow rate centrifugal compressor with CGCT in diffuser. Computational fluid dynamics (CFD) analysis is performed under stage environment in order to find the optimum location of the circumferential casing groove in consideration of stall margin enhancement and efficiency gain at design point, and the impact of groove number to the effect of this grooved casing treatment configuration in enhancing the stall margin of the compressor stage is studied. The results indicate that the centrifugal compressor with circumferential groove in vaned diffuser can obtain obvious improvement in the stall margin with sacrificing design efficiency a little. Efforts were made to study blade level flow mechanisms to determine how the CGCT impacts the compressor’s stall margin (SM) and performance. The flow structures in the passage, the tip gap, and the grooves as well as their mutual interactions were plotted and analysed.

1. Introduction
With the development of the modern engines, the compressor is required to have higher performance with fewer stages while satisfying ever stricter the safety and economical efficiency demands. From the viewpoint of compressor aerodynamics, deeper understanding of the stall process in rotor and system optimization is needed to provide better designs. A number of approaches have been proposed and adopted to enhance the compressor map width and overcome the problem of stall [1, 2].

Many efforts have been made in revealing the stall mechanism. Researchers have studied the flow structures near the endwall both experimentally and numerically to find possible stall triggers. Schlechtriem and Loetzerich [3], Hoffmann and Ballmann [4] and Yamada et al. [5] believe that the stall is triggered by the breakdown of the leading-edge blade tip leakage vortex (BTLV). However, Hah, Rabe and Wadia [6] find that for the forward swept rotor the vortex breakdown would not necessarily happen even if the compressor operates in a stalled condition, the shock oscillation and the BTLV oscillation induced by the shock-boundary layer interaction is a possible reason for the stall.
Those studies as well as some others’ results indicate that the stall process in the compressor is quite complicated and may be affected by many factors.

Casing treatment as a passive flow control method to extend the stable operating range for tip critical rotor has been used in practice for years. While it can give rise to improved stall margin, there can be an associated efficiency penalty. This is due to the lack of good knowledge on the mechanism of casing treatment, and the design often largely depends on experience. CGCT is a type of casing treatment which is simple and effective. Many research works have been done in this area. Rabe and Hah [7] conducted experimental and numerical investigations to study the fundamental flow mechanisms of CGCT in a transonic axial compressor and the influence on the compressor SM. Experimental results showed that circumferential grooves increased the SM of the compressor at the tested operating condition. Detailed investigation of calculated flow fields indicated that the grooves increased the SM by reducing the flow incidence angle on the pressure side of the leading edge and losses were generated by interaction between the main passage flow and flow exiting the grooves. Wilke and Kau [8] conducted numerical study on CGCT on a HPC front stage. The circumferential grooves are reported to suppress the extension of the BTLV and improve the stall margin. Shabbir and Adamiczky’s work [9] studied the physical mechanism of the CGCT with an analysis on axial momentum budget on a low speed rotor. Müller et al [10] conducted experimental and numerical investigations of CGCT on an axial single stage transonic compressor. Four different CGCT configurations were considered. The configuration with deep grooves and large coverage showed the best results on the stall margin improvement in both design and off design speed.

Recently, researchers also use circumferential grooves to expand the SM of centrifugal compressors. Liu et al. [11] designed two different CGCTs to enhance the SM of a centrifugal compressor. The results indicated that the first treated casing could extend the calculated mass flow range of the compressor by 10.5% and experimental case by 17.9% at 50% design speed with no additional large negative loss. Hu et al. [12] numerically studied the performance of a centrifugal compressor with CGCT. Their work indicated that CGCT could reduce the reverse flow of the tip leakage to the impeller inlet and delay the occurrence of stall. Gao et al. [13] performed three-dimensional (3-D) numerical simulation for a centrifugal compressor with CGCT to study the mechanism of stall margin improvement (SMI). They concluded that circumferential grooves could restrain the development of tip leakage vortex and delay the occurrence of stall.

Traditionally, in the above-mentioned literature works, the circumferential grooves are always placed along the casing wall of the rotor part of the compressor. However, the mechanisms of the stall and casing treatment are still not well understood. More efforts are needed not only on the mechanisms but also on providing some guidelines for the turbomachinery design. With rotating stall, which has recently been observed in vaned diffusers of centrifugal compressors by Spakovszky et al [14, 15]. Thus, in this article, the CGCT configuration [16, 17] is proposed and investigated as a control method to increase the SM. In this configuration, CGCT is placed along the shroud side of the diffuser passage to enhance the SM of the compressor. CFD analysis is performed under stage environment in order to find the optimum location of the CGCT in consideration of SM enhancement and efficiency gain at design point, and the influence of groove number in enhancing the SM of the compressor stage is studied. Also, the effect of CGCT to the compressor performance and near-tip flow structure was evaluated based on stall margin improvement.

2. Analysis Model
As previously explained, all experimental and numerical work is conducted on the single-stage, high speed, low flow rate centrifugal compressor. The design specifications of the impeller and diffuser are summarized in Table 1. The past numerical simulations indicate that there is a vortex in the diffuser passage of the compressor with smooth wall casing in near stall condition [18]. The overall performance and flow passage data obtained by experiment were referred for the validation of numerical modeling, which will be presented below.
Table 1. The design parameters of the compressor.

| Impeller                                | Diffuser                                |
|-----------------------------------------|-----------------------------------------|
| Blade inlet diameter (mm)               | The first vane inlet diameter (mm)      |
|                                         | 140                                     |
| Blade outlet diameter (mm)              | The first vane outlet diameter (mm)      |
|                                         | 240                                     |
| Tip clearance (mm)                      | The second vane inlet diameter (mm)     |
|                                         | 0.21                                    |
| Blade outlet diameter (mm)              | The second vane outlet diameter (mm)     |
|                                         | 12                                      |
| Number of rotor blades                  | Number of stator vanes                  |
|                                         | 9/9                                     |
| Rotation speed (rpm)                    | Design mass flow rate (kg/s)            |
|                                         | 30215                                   |
|                                         | 1.348                                   |

3. Numerical Method

3.1. Numerical Scheme

Three-dimensional flow calculations were performed in this article by using the software package NUMECA FINE/TurboTM to predict the performance of the compressor stage and to investigate the flow characteristics. The equations were discretized in space using a cell-centered finite volume formulation and in time using an explicit four-stage Runge-Kutta method. The one-equation Spalart-Allmaras turbulence model was adopted to evaluate the eddy viscosity. Local time stepping, implicit residual smoothing, and multi-grid techniques were used to reduce the computation cost.

The flow field inside the centrifugal compressor is inherently unsteady. Thus, in the state-of-the-art practice of CFD, unsteady simulations have become standard procedures in surge studies and can supply a precise prediction of the stall/surge point of the compressor. However, performing unsteady simulation is a really time consuming task, and unsteadiness had no significant effect on the performance of the low-loaded compressor. In this work, steady simulation is performed, and the flow communication between the rotor and stator uses a mixing plane method.

At the inlet, total pressure and total temperature with axial-flow direction is set as boundary condition. The ideal air is considered as working fluid. No-slip and no-heat transfer wall conditions are applied over all solid boundaries. Periodic conditions are applied for pitchwise boundaries other than the blade surfaces. Blade and wheel set as rotating parts, rotating speed as the design speed 30,215 RPM; wheel cover part set as stationary part. The mass flow rate is specified at the outlet of the compressor stage and is degressively reduced. When the stall point is impending, using the convergent solution with the lowest mass flow as the initial solution, one can move up the characteristics by decreasing the prescribed outlet mass flow in steps of 1g/s until the convergent solution no longer exists. The last converged or stable point is then referred to as the near-stall point.

3.2. Computational Grid

As the CGCTs are axially symmetrical, single passage simulations are enough to resolve all the
important flow mechanisms associated with casing treatment configuration. The calculation was done for a single passage of the centrifugal compressor. The computational grid of the impeller/diffuser stage is generated with the auto-grid generation software IGG/AutoGrid of NUMECA. The H&I topology structure was used in grid generation of impeller, proper encryption in the blade leading edge and trailing edge, the number of nodes is 1,223,952. Two series of vane diffuser using the default topology of NUMECA, the grid number is 202,223 and 198,695 respectively. The grids of circumferential groove were generated by ZR Effect function in Autogr5. The grids of circumferential groove and blade channel were connected by full no-matching boundary (FNMB). The grid node of circumferential groove casing treatment in three directions is 41, 37, 37, respectively, the total number of nodes is 56,129. The grid dependency analysis indicates that the computational grid is sufficient to obtain grid independent values for the figures of merit and can provide enough details for flow analysis. In order to properly evaluate the viscous fluxes at the walls, the minimum grid spacing away from the wall of the first node gives y+ to be less than 5 in the process of grid generating for all the computational cases and is sufficiently fine for the applied S-A turbulence model. Figure 1 displays the passage mesh of the impeller/diffuser stage, the diffuser passage with one circumferential groove.

3.3. Validation
The computational method was validated against the experimental data of the centrifugal compressor. The overall characteristics are shown with the corresponding experimental data in Figure 2. It can be seen from Figure 2 that simulation gives a qualitatively good agreement with experimental results. In numerical simulations, such as the insufficiency of the used turbulence model, the deviation of the real blade geometry, the simple hypothesis for ideal gas, without considering the leakage loss, wheel resistance loss, constancy hypothesis did not tally with the actual situation and other factors, the efficiency and pressure ratio of the experiment are lower than the calculated result, but the overall trend is apparent, the computational method is able to predict the flow with sufficient reliability.

4. Results and Discussion
4.1. Design of Circumferential Groove
In this paper, the circumferential groove slot positions were designed in the vane diffuser where stall inception occurred. Flow analysis of the solid casing stage showed that as the fluid diffuses in the impeller passage and interacts with the tip gap flow [16], it began to separate at the shroud side of the vaneless space and low momentum fluid accumulated near the shroud casing of the diffuser passage.

In order to determine the best slot position of the circumferential grooves, a series of radial location of circumferential grooves were designed along the wheel side of the diffuser and numerical simulations were carried out. The circumferential groove with radial width of 6mm and axial depth of 6mm. Figure 3 shows the slot location of circumferential grooves.
In order to quantitative the improvement of CGCTs to stall margin and the influence of stage efficiency at design point, stable working range $\Phi$, comprehensive improvement of stall margin $\Delta \Phi$, polytropic efficiency improvement at designed point $\Delta \eta_{pol}$ were evaluated in present work.

$$\Phi = \frac{Q_{m,des} - Q_{m,surge}}{Q_{m,des}} \times 100\%$$

$$\Delta \Phi = \frac{(Q_{m,surge})_{CT}}{(Q_{m,surge})_{SW}} \times \frac{(\varepsilon_{surge})_{CT}}{(\varepsilon_{surge})_{SW}} - 1$$

$$\Delta \eta_{pol} = \frac{(\eta_{pol,des})_{CT}}{(\eta_{pol,des})_{SW}} - 1$$

4.2. Impact of Groove Location on The Effect of Circumferential Groove

In Table 2, the term $R_{CGCT}$ indicates the radius of the leading edge of the treated grooves, as shown in Figure 3. Table 2 lists the specific calculation results of different radial location of circumferential grooves as CGCT No.1/2/3/4. Figure 4 shows the influence of circumferential groove at different radial locations.

Compared with the results of the smooth wall, it can be found from Table 2 and Figure 4 that the CGCT No.1 reached the largest stable working range of 47.48% and the most comprehensive stall margin improvement of 9.10%, but also caused the efficiency of the compressor design points reduced by 0.28%. The CGCT No.2 had smaller efficiency to reduce by 0.23%, and stable working range is 47.11%, 8.37% stall margin improvement. The CGCT No.3 and No.4 had less stall margin improvement and efficiency decreased more. In addition, it can be found from Table 2 that the circumferential grooves located in the leading edge and middle of the first vane diffuser can get a better expanding stability than the other circumferential grooves.

| Name               | No. 1 | No. 2 | No. 3 | No. 4 | SW |
|--------------------|-------|-------|-------|-------|----|
| $R_{CGCT}$ (mm)    | 128   | 136   | 144   | 156   | -  |
| $Q_{m,surge}$ (kg/s)| 0.708 | 0.713 | 0.733 | 0.735 | 0.771 |
| $\varepsilon_{surge}$ | 2.3686 | 2.3694 | 2.3937 | 2.3465 | 2.3642 |
| $\eta_{pol,des}$   | 0.8181 | 0.8185 | 0.8179 | 0.8177 | 0.8204 |
| $\Phi$ (%)         | 47.48%| 47.11%| 45.62%| 45.47%| 42.80%|
| $\Delta \Phi$ (%)  | 9.10% | 8.37% | 4.09% | 4.11% | -  |
Figure 4. Effects of different radial location circumferential grooves

4.2.1. Flow field at design point. Figure 5 presents the entropy contours at 95%-span in the diffuser passage at design point. The increase in the area of the low momentum fluid region near the diffuser casing wall due to the existence of circumferential grooves which effects the steady flow at design point contributes to the increase in the losses in the diffuser passage. It can be seen from the figure that, for the treated casing stages, the entropy value at the downstream region of the treated groove increases dramatically. For the region covered by the treated groove, the entropy value of the fluid also increases dramatically. The entropy value of CGCT No.3 increases more than CGCT No.2, which makes the CGCT No.3 efficiency decreases more at design point.

4.2.2. Flow field at near-stall point. Figure 6 gives flow details and entropy distribution in the circumferential casing grooves. The stage with CGCT No.3 shows the higher entropy value and consequently the lower efficiency. As for the CGCT, there is one single regular vortex in the groove; the core of the vortex almost locates at the centre of the groove. For the treated casing diffusers, the existence of the CGCT supplies a recirculation passage for the low momentum fluid near the diffuser casing. The low momentum fluid is sucked into the groove and transports along the circumferential

![Diagram](image1)

Figure 5. Entropy distribution at 95%-span in the diffuser passage at design point

![Diagram](image2)
Figure 6. Relative velocity vector and entropy distribution in CGCTs at near-stall point

Figure 7. Computational grid of diffuser passage with different number grooves

direction in the groove. Thus, a single regular vortex locating near the centre of the grooves helps to improve the flow status in the stage, and contributes to the improvement of the stall limit of the stage.

4.3. Impact of Groove Number on The Effect of Circumferential Groove

Table 3 lists the specific calculation results of different numbers of circumferential grooves as CGCT-1 (No. 2), CGCT-2 (No.1 and No.2) and CGCT-3 (No.1, No.2 and No.3). Figure 7 presents the computational grid of diffuser passage with different number CGCTs. Figure 8 shows the influence of circumferential grooves with different number.

Compared with the results of the smooth wall, it can be found from Table 3 and Figure 8 that when the circumferential groove number increased from 1 to 2, stable working range and comprehensive stall margin improvement enhanced with static pressure ratio fell and the efficiency decreases obviously. When the circumferential groove number increase from 2 to 3, stable working range continue to increase, while the increasing amplitude is reduced, the comprehensive stall margin improvement decreased, and the static pressure continue to decline, efficiency decreases more.

Table 3. Results of different groove number of circumferential grooves

| Name      | CGCT-1  | CGCT-2 (No.1/No.2) | CGCT-3 (No.1/No.2/No.3) | SW |
|-----------|---------|--------------------|--------------------------|----|
| Q_{m,surge} (kg/s) | 0.713   | 0.701              | 0.699                    | 0.771 |
| ε_{surge}  | 2.3694  | 2.3468             | 2.3312                   | 2.3642 |
| η_{pol,des} | 0.8185  | 0.8148             | 0.8063                   | 0.8204 |
| Φ          | 47.11%  | 48.00%             | 48.15%                   | 42.80% |
| ΔΦ         | 8.37%   | 9.18%              | 8.76%                    | -    |
| Δη_{pol}   | -0.23%  | -0.68%             | -1.72%                   | -    |

Table 3. Results of different groove number of circumferential grooves
4.3.1. Flow field at design point. Figure 9 and Figure 10 examine the relative velocity vector and entropy distribution at inlet of CGCTs at design point for the treated casing stages with 2 and 3 grooves. It can be seen that, as the number of grooves increases, the flow status at the inlet cross-section plane of the same groove has little difference in velocity vector. After the low momentum fluid is sucked into the groove, the fluid expresses as circumferential transportation from the pressure side of the diffuser vane to the suction side near the diffuser casing wall. As the number of grooves increases, the entropy at the inlet of CGCT-3 has obviously increase than CGCT-2, and the losses in the groove increase, which makes the efficiency of CGCT-3 decreased at design point.

4.3.2. Flow field at near-stall point. Figure 11 and Figure 12 present the axial velocity component distribution at inlet of CGCTs for the treated casing stages with 2 and 3 grooves at near-stall point. In the following figures, the positive axial velocity indicates that the fluid outflows from the groove, while the negative axial velocity indicates that the fluid inflows into the groove. It can be seen that, for the same groove No.1 and No.2, as the number of grooves increases, the extents of the positive and negative velocity core regions shrink obviously. The suction-reinjection capacity of the groove to the low momentum fluid near the diffuser casing wall decreases, and at the inlet cross-section plane of the groove, the flow status deteriorates. Comparing with the second groove, the axial velocity at the inlet cross-section plane of the third groove decreases obviously. It can be seen that, as the number of grooves increases, the suction-reinjection capacity of the single groove to the low momentum fluid decreases, but the stable working range increase with the whole capacity increase.

From the above analysis, we can conclude that the existence of the CGCTs supplies a recirculation passage for the low momentum fluid near the diffuser casing wall at near-stall operating conditions. The low momentum fluid is sucked into the treated casing groove and transports along the circumferential and streamwise directions in the groove. As only a part of the fluid reinjects into the main flow, the area of the low momentum fluid region shrinks and the velocity increases. Thus, the
initiation of stall is delayed, and the overall contribution of each groove to SM gain depends on the compound effect of suction-reinjection.

Figure 11. Axial velocity component distribution at inlet of CGCT-2 at near-stall point

Figure 12. Axial velocity component distribution at inlet of CGCT-3 at near-stall point

5. Concluding Remarks
To evaluate the effects of radial locations and the number of CGCTs in vaned diffuser on the stall margin improvement (SMI) of a centrifugal compressor, steady three-dimensional Navier-Stokes flow simulations were conducted in this paper. The conclusions are summarized as follows:

(1) When the CGCTs is applied at the endwall of the diffuser, suitable position of circumferential groove can effectively expand the SM of the compressor, and the efficiency has fallen a little at the design point.

(2) The investigation based on CGCT-2, which consists of two circumferential grooves, indicates that CGCT-2 can generate maximum SMI, and CGCT-3 including three grooves provides more SMI than CGCT-1 including only the first groove of CGCT-2. The compressor with CGCT No.3 including only the third groove of CGCT-3 gains minimum SMI.

(3) The stage efficiency at the design point decreases as the groove number increases. The near-stall mass flow moves to a lower value and the SMI increases when increasing the groove number from 1 to 2. However, the SMI decreases when the groove number is increased to 3. The overall contribution of each groove to SMI depends on the compound effect of suction-reinjection.

(4) The evaluation based on stall margin improvement showed the optimal position for the groove to be located was indicated to exist near the leading edge of the diffuser, and a combination of geometry parameters and number of circumferential grooves that will maximize both SMI and efficiency.

Nomenclature
CGCT= circumferential groove casing treatment
SMI = stall margin improvement
\( Q \) = quantity of flow
\( \varepsilon \) = static pressure ratio
η = efficiency
Φ = stable working range
Δ Φ = comprehensive improvement of stall margin
Δ η = efficiency improvement

Subscripts
m = mass
surge = surge point condition
pol = polytropic
des = design point
SW = smooth wall case
CT = casing treatment case

Acknowledgements
Financial support for the work presented is provided by the National Key Basic Research Program of China (Project No. 2012CB026000), this support is greatly appreciated.

References
[1] Ishida M, Surana T and Sakaguchi D 2005 Trans ASME, J. Turbomach. 127(1) 76-83.
[2] Choi K J, Kim J H and Kim K Y 2010 ASME Paper No. GT2010-22396.
[3] Schlechtriem S and Loetzerich M 1997 ASME Paper No. GT97-41.
[4] Hoffman W H and Ballman J 2003 ISABE Paper No. 2003-1223.
[5] Yamada K, Furukawa M, Inoue M and Funazaki K 2003 IGTC Paper No. 2003-095.
[6] Hah C, Rabe D C and Wadia A R 2004 ASME Paper No. GT2004-53867.
[7] Rabe D C and Hah C 2002 ASME Paper No. GT2002-30641.
[8] Wilke I, Kau H P 2003 ASME Paper No. GT2003-30642.
[9] Shabbir A and Adamiczky J J 2005 ASME J. Turbomach. 127 708-717.
[10] Müller M W, Schiffer H P and Hah C 2007 ASME Paper No. GT-2007-27365.
[11] Liu B, Wang Q W and Chen Y Y 2009 ISABE Paper No. ISABE2009-1268.
[12] Hu L J, Yang C, Qi M X and Ma C C 2009 Chinese Journal of Mechanical Engineering 45(7) 138-144.
[13] Gao P, Chu W L, Wu Y H and Deng W J 2007 Journal of Propulsion Technology 28(4) 383-387.
[14] Spakovszky Z S and Roduner C H 2009 ASME J. Turbomach. 131(3) 031012.
[15] Everitt J N and Spakovszky Z S 2013 ASME J. Turbomach. 135(1) 011025.
[16] Ma Y, Xi G and Wu G K 2011 ASME Paper No. GT2011-46015.
[17] Xi G, Ma Y, Wu G K, Zhang K Y, Xiao W and Mou Z J 2013 Journal of Power and Energy 227(4) 464-478.
[18] Chen X F, Qin G L and Ai Z J 2015 IOP Conference Series: Materials Science and Engineering 90(1) 012043.