EXTENDED METHODOLOGY FOR ANALYSING THE FLOW MECHANISMS INDUCED BY CASING TREATMENT IN COMPRESSOR

G. Legras\textsuperscript{1} - N. Gourdain\textsuperscript{2} - I. Trébinjac\textsuperscript{3}

\textsuperscript{1}SNECMA - SAFRAN GROUP Aerodynamics Methods and Tools Development Department, Rond Point René Ravaud, 88550, Moissy-Cramayel, France, legras@cerfacs.fr
\textsuperscript{2}CERFACS Computational Fluid Dynamics Team 42 av. Gaspard Coriolis 31057, Toulouse, France, gourdain@cerfacs.fr
\textsuperscript{3}LMFA, UMR 5509 Ecole Centrale de Lyon / UCBLyon I / INSA 69134 Ecully, France, isabelle.trebinjac@ec-lyon.fr

ABSTRACT

The present paper develops a methodology to uncover the physical mechanisms of casing treatments in compressors. The current method is a generalization of the approach originally proposed by Shabbir and Adamczyk (2005). It is based on a budget analysis of the Navier Stokes set of (un)steady equations allowing the understanding and the quantification of complex flow mechanisms such as those induced by casing treatments. The second issue of the paper focuses on the application of the current approach to better understand the physical effects induced by circumferential grooves on a transonic compressor. The well-known test case NASA Rotor 37 is numerically tested for the application of the casing treatment. Results of budget analysis of the axial and radial momentum reveal that the tip flow of the smooth casing is strongly influenced by its radial velocity component. It is mainly due to (i) the tip leakage flow, which is easily controlled by the grooves, and (ii) the shock wave with concave shaped casing.

NOMENCLATURE

Symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>$[A_r, A_\theta, A_z]^T$ [m$^2$]</td>
</tr>
<tr>
<td>$E_r$</td>
<td>[J.kg$^{-1}$]</td>
</tr>
<tr>
<td>$F_{cen}$</td>
<td>[N.kg$^{-1}$]</td>
</tr>
<tr>
<td>$F_{cor}$</td>
<td>[N.kg$^{-1}$]</td>
</tr>
<tr>
<td>$F_{k2}$, $F_{k4}$</td>
<td>2\textsuperscript{nd} and 4\textsuperscript{th} order numerical scalar artificial dissipation fluxes</td>
</tr>
<tr>
<td>$h$</td>
<td>[m]</td>
</tr>
<tr>
<td>$H$</td>
<td>[m]</td>
</tr>
<tr>
<td>$P_s$</td>
<td>[Pa]</td>
</tr>
<tr>
<td>$q$</td>
<td>$[q_r, q_{\theta}, q_z]^T$ [J.m$^{-2}$.s$^{-1}$]</td>
</tr>
<tr>
<td>R</td>
<td>numerical modelling residual</td>
</tr>
<tr>
<td>$R_{pi}$</td>
<td>[-]</td>
</tr>
<tr>
<td>r, $\theta$, z</td>
<td>[-]</td>
</tr>
<tr>
<td>t</td>
<td>[s]</td>
</tr>
<tr>
<td>$V$</td>
<td>[m$^3$]</td>
</tr>
<tr>
<td>$V$</td>
<td>$[V_r, V_\theta, V_z]^T$ [m.s$^{-1}$]</td>
</tr>
<tr>
<td>$W$</td>
<td>$[W_r, W_\theta, W_z]^T$ [m.s$^{-1}$]</td>
</tr>
</tbody>
</table>

Greek letters

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\eta_s$</td>
<td>[-] isentropic efficiency</td>
</tr>
<tr>
<td>$\gamma$</td>
<td>[-] specific heat ratio</td>
</tr>
<tr>
<td>$\mu$</td>
<td>[Po] molecular dynamic viscosity</td>
</tr>
<tr>
<td>$\omega$</td>
<td>[rad.s$^{-1}$] rotation speed</td>
</tr>
<tr>
<td>$\phi$</td>
<td>[-] normalized massflow</td>
</tr>
<tr>
<td>$\rho$</td>
<td>[kg.m$^3$] density</td>
</tr>
<tr>
<td>$\alpha$</td>
<td>[$^\circ$] absolute flow angle</td>
</tr>
<tr>
<td>$\tau$</td>
<td>[Pa] sum of the viscous and turbulence stress tensor</td>
</tr>
</tbody>
</table>

Acronyms

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>CT</td>
<td>Casing Treatment</td>
</tr>
<tr>
<td>SW</td>
<td>Smooth Wall</td>
</tr>
<tr>
<td>(U)RANS</td>
<td>(Unsteady) Reynolds Averaged</td>
</tr>
</tbody>
</table>

INTRODUCTION

Today, it is mandatory for compressor designers to improve performance in terms of efficiency and operating range characterized by the stall margin. One of the main difficulties encountered in this process is that compressor stall is not always controlled through normal aerodynamic design.
Thus stall prevention techniques must be used and one promising technology known to bring substantial stability in the rotor tip region is casing treatment (Greitzer, 1979). This kind of technology consists of slots or grooves within the rotor casing. Many experimental and numerical studies have been performed that attempt to uncover the physics behind the improvement in stall margin: Wilke et al. (2005) on slot-type, Houghton and Day (2009) on circumferential casing grooves and Hathaway (2002) on self-recirculating bridge. However this understanding is still not complete since experimental measurements in the near casing region are elaborate. Moreover, numerical results post-processing using 2D or 3D views hardly allow a quantification and can cause in some cases a mis-understanding of the flow mechanisms. Therefore, casing treatment design requires analysis methods that need to be as easy as possible to handle for aeroengine designers. In this context, Shabbir and Adamczyk (2005) proposed an approach based on a budget analysis of the steady axial momentum equation close to the rotor casing. This methodology provides further insight into the flow mechanisms relevant to compressor stability systems. In fact, their paper shows the approach is innovative to quantitatively ascertain the influence of a design on the near casing flow, and providing guidance on grooves design along the axial direction. However, Shabbir and Adamczyk methodology is restrained to the knowledge of changes in the balance of steady axial momentum equation while casing treatment interactions with the main flow are strongly complex, thus requiring information that can provide the overall Navier-Stokes equations.

In order to further understand and quantitatively diagnose complex flow mechanisms such as those induced by casing treatments, the current paper presents generalization of the methodology originally proposed by Shabbir and Adamczyk (2005). The new method calculates the budget analysis of the Navier-Stokes set of (un)steady equations. In this paper, the method are fully described and used to investigate the flow mechanisms induced by circumferential casing grooves on the well-known transonic compressor NASA Rotor 37. Results are based on numerical 3D steady calculations (previously presented in Legras et al., 2010). In order to apprehend this novel approach, the budget analysis is restrained to the steady axial momentum equation and steady radial momentum equation.

**DESCRIPTION OF THE FLOW ANALYSIS METHODOLOGY**

**Equations**

Following the way proposed by Shabbir and Adamczyk (2005), a control volume fixed in time and located in the near casing flow is retained. This will provide a quantitative understanding of the relevant fluid mechanisms associated with smooth and treated casing configurations. Thus, the balance between the various terms which appear in the Navier-Stokes equations is analyzed based on its finite volume formulation. For simplicity reasons, the equations are considered in the relative rotor frame and in cylindrical coordinates. Using the divergence theorem, the averaged Navier-Stokes equations in integral form can be written by:

\[ V \int_{\partial \Omega} \frac{\partial Q}{\partial t} = - \oint_{\partial \Omega} \left[ B \, dA_r + C \, dA_\theta + D \, dA_z \right] + V \, T \]  

where \( A_r, A_\theta \) and \( A_z \) are the projection areas of the control volume, \( V \) the constant volume of the control domain, \( Q \) the conservative variables, \( B, C \) and \( D \) flux vectors resulting from the development of the advective and diffusive fluxes and \( T \) the forces per unit volume (usually named source terms). These vectors are written as follows:

\[
Q = \begin{bmatrix} \rho & \rho W_r & \rho W_\theta & \rho W_z & \rho E \end{bmatrix}, \quad
B = \begin{bmatrix}
\rho W_r \\
(\rho W_r^2 + P_r) - \tau_{rr} \\
\rho W_\theta W_r - \tau_{r\theta} \\
(\rho E_r + P_r - \tau_{rr}) W_r - (W_\theta \tau_{r\theta} + W_z \tau_{rz} + q_r)
\end{bmatrix}, \quad
T = \begin{bmatrix}
0 & \rho W_\theta^2 - P_\theta - \tau_{\theta\theta} + 2 \rho \omega W_\theta + \rho \omega^2 r & 0 \\
\rho W_\theta W_r - \tau_{r\theta} & 0 & -2 \rho \omega W_r \\
0 & \rho \omega^2 r W_\theta & 0
\end{bmatrix}
\]

(2)
The stresses in Eq. 2 include both viscous and Reynolds stresses. In the same manner, the heat flux takes into account the heat flux and the enthalpy turbulent diffusion flux. This last term and the Reynolds stresses are both approximated by an eddy viscosity model.

**Numerical Resolution**

Since the objective is to understand the balance of the various terms by using numerical CFD results, the semi-discretised in space for un-coupled time/space integration formulation of the Eq. 1 is considered. For an individual basic hexahedral cell, this formulation is written as follows:

\[
\frac{\partial Q}{\partial t} = -\frac{1}{V} \left[ \sum_{l=1}^{6} F \cdot N_{\Sigma_l} - V T \right] = -\frac{1}{V} R(Q)
\]

where \( l \) designates the \( l^{th} \) face bounding the hexahedral cell, \( F \) the numerical approximation of the exact flux (including the below tensors \( B, C \) and \( D \)), \( N_{\Sigma_l} \) the external (non unit) normal to the face \( \Sigma_l = [A_{rl}, A_{\theta l}, A_{zl}]^T \) and \( R \) the numerical modelling residual of variable \( Q \).

In order to access the information of all individual terms, a computation post-processing tool has been developed. Since the volume does not depend on time, the source terms \( T \) can be directly determined using Eq. 2. Concerning the time derivative terms, the choice is done to determine those terms at instant \( n \) by computing the opposite of the numerical modelling residual at the same instant:

\[
V \frac{\partial Q^n}{\partial t} = -R(Q^n)
\]

Therefore, only the convective fluxes have to be computed (here with a 2nd-order centered Jameson scheme (Jameson, 1981)).

Since most of CFD codes solve the Navier-Stokes equations in the cartesian reference frame, a specific treatment must be done to recover terms of Eq. 2 in cylindrical coordinates. To do so, the fluxes constitutive of terms in Eq. 2 are build up through projection of the cartesian fluxes into the cylindrical reference frame. This strategy, instead of applying a cylindrical spatial scheme, permits to ensure the same numerical modelling residual than in the cartesian equations. Equation 5 presents the transformation matrix \( P \) for rotation by an angle of \( \theta \) over the longitudinal direction (i.e. \( z \)). Equation 6 recalls the expressions for projections from cartesian to cylindrical coordinates of the face areas \( A \), the relative velocity vector \( W \), the stress tensor \( \tau \) and the heat flux vector \( q \).

\[
P = \begin{bmatrix} \cos \theta & -\sin \theta & 0 \\ \sin \theta & \cos \theta & 0 \\ 0 & 0 & 1 \end{bmatrix}
\]

\[
A_{r\theta z} = P^{-T} \cdot A_{xyz} \quad W_{r\theta z} = P^{-T} \cdot W_{xyz} \quad \tau_{r\theta z} = P^{-T} \cdot \tau_{xyz} \cdot P \quad q_{r\theta z} = P^{-T} \cdot q_{xyz}
\]

For sake of clarity, the nomenclature used by Shabbir and Adamczyk (2005) is taken. The operator \( \Delta() = \sum_{l=1}^{6}() \) is introduced and characterizes the balance of flux on an individual basic cell. For example, the term \( \Delta(\rho W_r A_r) \) appearing in the axial momentum equation corresponds to the transport of the axial momentum across the radial faces of a basic grid cell.
The objective is to understand the balance of the various terms by using numerical results. Thus, it is obvious that the control volume retained is based on the meshing used. In consequence, the current approach is extended to a control volume composed of multiple grid cells. Therefore, the operator \( \sum_{r,\theta,z}() = \sum_{r,\theta,z}() \) is introduced and realizes the cumulative sum on each individual grid cell. For example, the axial momentum equation can be written as:

\[
\frac{1}{V} \sum_{r,\theta,z} \Delta \left( \rho W_z W_r A_r \right) + \sum_{r,\theta,z} \Delta \left( \rho W_z W_\theta A_\theta \right) + \sum_{r,\theta,z} \Delta \left( \rho W_z^2 A_z \right) + \sum_{r,\theta,z} \Delta \left( P_s A_z \right) + \sum_{r,\theta,z} \left( F_{k2} - \sum_{r,\theta,z} \Delta \left( \tau_{rz} A_r \right) - \sum_{r,\theta,z} \Delta \left( \tau_{\theta z} A_\theta \right) - \sum_{r,\theta,z} \Delta \left( \tau_{zz} A_z \right) \right) = \frac{1}{V} \rho W_z \frac{\partial \rho W_z}{\partial t}
\]

where \( F_{k2} \) and \( F_{k4} \) correspond respectively to the 2nd and 4th numerical scalar artificial dissipation fluxes. These have been added to the equation due to employment of 2nd-order Jameson centered space-discretization scheme (Jameson, 1981) in the numerical simulations. This allows the quantification of artificial dissipation fluxes on the balance of the axial momentum equation. Note that each term is homogeneous to a force per unit volume (\([\text{kg.m}^{-2}.\text{s}^{-1}]\)).

**Application of the Current Approach**

**On Steady Flows**

In the case of steady flow problems, time derivative terms are null leading to a balance of the advective and viscous fluxes and forces per volume unit. In practice, the precision of this equilibrium depends on the value of the numerical residual \( R \). This tends to reach zero at convergence according to the machines errors. Assuming that the numerical result is correctly converged, Eq. 3 can be simplified as follows:

\[
R \left( Q \right) \approx 0 \iff \sum_{i=1}^{6} F \cdot N_{\Sigma_i} - V T_O \approx 0
\]

**On Unsteady Flows**

Even if the current paper presents steady flow analysis of the NASA Rotor 37, it is instructive to dwell on application of the current approach on unsteady flow problems. In such a case, time derivative terms need to be determined since it can strongly impact the equilibrium of the equations. Those are taken equal to \(-R \left( Q \right)\), which is consistent only if the result analysed is a time consistent solution (usually a periodic solution) converged in time step.

The current approach can be applied both on unsteady and time-averaged solution allowing access of specific information of the numerical modelling residual. In fact, assuming that a flow variable \( Q \) can be decomposed into its mean term \( \overline{Q} \) plus its deterministic components \( Q' \) (stochastic components \( Q'' \) are neglected), i.e. \( Q = \overline{Q} + Q' \), and by replacing each variable in the Eq. 1, the residual operator can be written as:

\[
\overline{R} \left( Q \right) = R \left( \overline{Q} \right) + \overline{R} \left( Q' \right) = 0
\]

where \( \overline{R} \left( Q \right) \) denotes the time-averaged balance of equation and tends to zero since the unsteady problem is periodic in time. \( R \left( \overline{Q} \right) \) designates the residual applied to time-averaged flow field. \( \overline{R} \left( Q' \right) \) indicates the time-averaged effect of the unsteadiness (also called Lumped Deterministic Source Terms by Ratzlaff et al. 2008). Therefore, Eq. 9 shows that this last term can be found exactly by computing \(-R \left( Q \right)\). Specific terms, as for example \( \sum \Delta \left( \rho W_z W_r A_r \right) \) corresponding to the time-averaged effect of the unsteady radial transport of axial momentum, can be accessed by identification.
CASING GROOVES ON TRANSONIC AXIAL COMPRESSOR AND APPLICATION OF THE CURRENT APPROACH

This section is dedicated to the numerical application of the methodology previously developed. In this current paper, it is chosen to apply the approach only on a numerical steady flow problem. Therefore, circumferential casing grooves on transonic axial compressor are investigated in order to reach a comprehensive description of the change in flow topology.

NASA Rotor 37 with Circumferential Casing Grooves

The transonic compressor used for the current study is the well-known experimental compressor NASA Rotor 37 (Fig. 1) designed and tested at NASA Lewis Research Center (Reid and Moore, 1978-1980). The main characteristics are recalled Table 1. More details about the compressor design, extensive results and comparisons of numerical and experimental data may be found in the following references: Suder and Celestina (1996), Chima (1996), Denton (1996), Dunham (1998). The NASA Rotor 37 has been chosen since it is known for the occurrence of blade tip located rotating stall phenomena at operating close to surge (Wilke et al., 2005). Note that the test case does not include any casing treatment (i.e. experimental data are for the untreated configuration). Casing grooves implementation is only investigated through numerical simulations.

The casing treatment geometry is taken from Legras et al. (2010) and consists of 6 circumferential slots (Fig. 6 and 7). Each groove is 3 mm wide (10.9% of rotor tip axial chord) with a gap of 1.5 mm between each cavities. The height to width ratio is 0.7:1. The first groove starts at around \(x/C = 5\%\) of the tip chord upstream of the rotor tip leading edge. Concerning the design, the objective were not to optimize the casing treatment geometry but to reach a comprehensive description of the change in flow properties generated by grooves. Therefore, it was found interesting to investigate the impact of a slot that surmounts the blade tip leading edge part, even if its usefulness is not well clear. The methodology previously developed aims at clarifying this point.

Description of the Numerical Method

As casing grooves are axisymmetric, they can be considered as a continuing circumferential modification of the casing and then be computed similarly to smooth wall configuration in a conventional steady-state approach. Thus, two sets of steady calculations were run with and without the grooves on the isolated rotor. Numerical simulations are carried out using the elsA software developed by ONERA and CERFACS (Cambier and Veuillot, 2008) and validated on the NASA Rotor 37 (Castillon

<table>
<thead>
<tr>
<th>Number of blades</th>
<th>36</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tip radius at leading edge</td>
<td>252 mm</td>
</tr>
<tr>
<td>Aspect ratio</td>
<td>1.19</td>
</tr>
<tr>
<td>Hub-tip radius ratio</td>
<td>0.70</td>
</tr>
<tr>
<td>Tip solidity</td>
<td>1.288</td>
</tr>
<tr>
<td>Tip clearance height</td>
<td>0.356 mm</td>
</tr>
<tr>
<td>Rotation speed</td>
<td>17188.7 rpm</td>
</tr>
<tr>
<td>Tip speed</td>
<td>457 m/s</td>
</tr>
<tr>
<td>Total pressure ratio</td>
<td>2.106</td>
</tr>
<tr>
<td>Adiabatic efficiency</td>
<td>0.877</td>
</tr>
<tr>
<td>Design mass flow</td>
<td>20.188 kg.s(^{-1})</td>
</tr>
<tr>
<td>Choked mass flow</td>
<td>20.93 ± 0.14 kg.s(^{-1})</td>
</tr>
</tbody>
</table>

Table 1: Design parameters of the NASA Rotor 37 smooth configuration (Dunham, 1998).
et al., 2002). The elsA code solves the Favre-Reynolds-averaged Navier-Stokes equations on multi-block structured meshes using a cell-centered finite-volume approach. The code also allows the use of the Chimera method dedicated to complex geometries typically generated by technological effects (Castillon et al., 2010). Computations were run with a 2nd-order centered Jameson scheme (Jameson, 1981) for the estimation of convective fluxes. The two-equation model proposed by Wilcox based on a \( k - \omega \) formulation was used to model turbulence. Moreover, the compressor flow is assumed to be fully turbulent \( (Re \approx 5 \times 10^6) \). Other compressor modelling details are given in Legras et al. (2010).

The flow domain is discretized with a low-Reynolds multi-block structured approach. The smooth wall configuration uses an O3H meshing strategy with a total of 1.56 million grid points. The mesh is characterized by 89 grid points in the span-wise direction with 25 grid points in the tip-clearance gap. The casing treatment configuration is meshed using the smooth wall mesh at which H Chimera blocks modelling the casing grooves have been added (2.26 million points).

**Preliminary Considerations on the Simulations**

**Validation of the smooth casing simulation model**

To validate the simulations, a number of flow characteristics have been compared to the experimental data for NASA Rotor 37 with smooth casing. The measuring locations are shown in Fig. 1. Measured and calculated performances are presented in Fig. 2 at design speed over the experimental operating range. Mass flow rate are normalized by the experimental choked mass flow \((20.93 \pm 0.14 \text{ kg.s}^{-1})\). The pressure ratio - mass flow curves show a fairly good agreement between numerical and experimental results of the smooth wall configuration. However, the efficiency curve resulting from the simulation is clearly shifted which leads to underestimate the efficiency whatever the mass flow.

Comparing the radial distributions of pitch-wise averaged total pressure (Fig. 3(a)) at the rotor outlet (station 4), it can be seen that measured and simulated profiles are almost similar. Light differences mainly occur near the outer casing that have also been observed by other authors who numerically investigated the NASA Rotor 37 (Wilke et al., 2005). Moreover, one can noticed that near the hub region the simulation fails to predict the deficit of the stagnation pressure suggested by the measurements. Hah and Loellbach (1999) attribute this deficit to important corner stall. Shabbir et al. (1997) argue that the hub disk leakage flow, emanating from the axial gap existing between the fixed and rotating parts of the hub upstream of the blade, is responsible. Recent work done by Castillon et al. (2010) with the elsA software confirms this last result. They show that this leakage merges with streamlines coming from the hub boundary layer. The streamlines migrate on the suction side of

---

![Graph](image-url)

Figure 2: Predicted NASA Rotor 37 operating characteristics at design speed.
the blade due to the pressure gradient across the blade passage, amplifying corner stall, thereby increasing aerodynamic losses and reducing the compression rate. The present authors believe that the difference in performance (Fig. 2(b)) is attributed to not taking into account the axial gap at hub in the present modelisation. The plot showing blade-to-blade distributions of the relative Mach number at 20% chordwise and 95% spanwise location (Fig. 3(b)) affirms the good agreement between simulation and measurements of the NASA Rotor 37 with smooth casing. Therefore, it was considered that the numerical model is able to reveal the overall flow mechanisms occurring near the outer casing, which is essential for the objective of the present work aiming at understanding the local influence of casing grooves.

(a) Total pressure ratio (span-wise, station 4). (b) Relative Mach number (pitch-wise, 95% and 20% blade chord, station 2).

Figure 3: Comparison of the measured and simulated distributions at peak efficiency and near stall operating points.

Flow mechanism near stall operating point for the smooth casing configuration

Fig. 4(a) shows the relative helicity ($= \mathbf{W} \cdot \text{rot}\mathbf{W}$) flow field at blade tip of the simulated smooth wall configuration near stall operating point ($\phi \approx 0.93$). The helicity flow variable helps on revealing details of tip clearance flow field such as vortices structures. This plot confirms the main conclusions of previous works (CFD study by Chima (1996) and experimental study by Suder and Celestina (1996)), that tip wall region is dominated by a strong tip leakage flow. This last rolls-up into a streamwise vortex that develops close to the leading edge on the blade suction side. The vortex then linearly carried out in the channel by the main flow until it interacts with the blade passage shock at mid-span between pressure and suction side. At near stall operating condition, the vortex core structure abruptly expands downstream of the shock which is highlighted by the deficit of helicity. This dramatic change leads to a large blockage effect that deflects the main flow toward the neighboring channel, marking the last stable numerical operating point. This phenomenon has already been identified and analyzed by other authors in former studies (Chima, 1996; Wilke et al., 2005).

Simulated casing treatment for the NASA Rotor 37

Numerical results of the casing treatment configuration show almost insignificant differences with the smooth wall case in terms of performance (Fig. 2) and distributions (Fig. 3). Those low differences are all the more surprising that important radial flow exchanges occur between slots and the main flow (Fig. 5). Indeed, grooves generate a radial transport of low energy fluid from casing boundary layer to the grooves. One notices that the flow recirculation mainly occurs for the 2nd to the 4th grooves. A good representation of the main impact of grooves in terms of losses is obtained through the entropy
level. Fig. 6 shows the relative difference of entropy between the two simulations. Results reveal insignificant difference except near the tip gap region between the previous 2nd to the 4th grooves, which is highlighted by higher entropy level (+0.5% of relative difference). In addition to the mixing loss induced by the radial flow exchanges, it can be observed that this higher entropy region corresponds also to the zone of vortex/shock interaction, which is slightly more intense than the smooth wall case. Relative helicity flow field at i.e. $\phi \approx 0.93$ and presented in Fig. 4(b) confirms the occurrence of a strong vortex breakdown. Moreover, the plot highlights the formation of secondary tip leakage vortices, resulting from the grooves/leakage flow interaction and contributing to higher mixing loss (Legras et al., 2010). Furthermore, the main tip leakage vortex has a lower trajectory angle relative to the blade tip chord line by $2^\circ$ compared to smooth case. According to Legras et al. (2010), this is due to lower blade tip loading. Despite that this result can itself improves the stable operating range (according to Rabe et Hah, 2002), the strong vortex/shock interaction creates as much blockage as in the smooth wall case at $\phi \approx 0.93$, thus limiting the performance of the simulated grooved configuration.

![Figure 4: Relative helicity flow field noted at 96% of the rotor blade span for the near stall operating condition $\phi \approx 0.93$ (Legras et al., 2010).](image)

![Figure 5: Radial velocity normalized by the blade tip velocity at the blade tip rotor span.](image)

![Figure 6: Meridian views of the relative difference of entropy level between the smooth (ref.) and grooved casing calculations at near smooth casing stall point.](image)

It is interesting to note that nearly all numerical studies that investigate casing treatment for the NASA Rotor 37 present a relative insensitivity of the slots on the performance (circumferential grooves: Haixin et al. (2005), Huang et al. (2008), Beheshti et al. (2004); semi-circular slots:
Wilke et al. (2005)). However, it is interesting to better understand the reason why implementation of casing treatment is difficult in the NASA Rotor 37. The application of the methodology previously developed on the near casing flow region can be useful to ascertain the flow mechanisms and quantifies fluxes (i.e. forces acting on the flow).

**Flow Mechanisms Analysis according to the Current Approach**

In order to get further insight into the grooves flow mechanisms, the stall point defined as the last stable point experimentally measured is chosen for comparisons between smooth and grooved casings (i.e. \( \phi \approx 0.93 \)). The following paragraphs investigate the balance of the steady axial momentum equation (Eq. 7 with \( R(\rho W_z) \approx 0 \)) since the pressure rise across the rotor can be explicitly derived from it. At the same time, the steady radial momentum equation (see Eq. 1) is analyzed because of its formulation close to the expression of radial equilibrium (differentiated by the frame of reference, respectively relative and absolute).

**Definition of the control volume**

Using numerical results of the isolated rotor with and without casing grooves, Eq. 7 is applied on a control volume surrounding the rotor blade tip flow field (Fig. 7). It is circumferentially delimited by the rotor blade pitch, axially extending just upstream and downstream of the rotor blade tip, and radially bounded by the 5th/25 to the 20th/25 grid layers modelling the rotor tip clearance. Therefore, the radial thickness of the control volume does not go to zero due to the Chimera technique in the grooved configuration. In fact, the two last radial layers attached to the casing are used for interpolation between grooves and the blade passage blocks. These are not taken into account in the control volume since they can mislead the Navier-Stokes equations equilibrium.

**Global Flow Mechanisms: 0-D Analysis**

Smooth and grooved casings results of the steady axial momentum and radial momentum balances are presented respectively in Fig. 8(a) and Fig. 8(b). This kind of histogram provides a macroscopic view of the forces acting on the control volumes.

Before discussing on the physical analysis, it is instructive to comment on the magnitude of the numerical terms of the balance. Results show that the numerical modelling residuals \( R(\rho W_z) \) and \( R(\rho W_r) \) tend to zero leading to a good precision of the balance of the equation. Moreover, it can be observed that scalar artificial viscosity fluxes \( (F_{k2} \text{ and } F_{k4}) \) are insignificant in spite of the presence of a strong shock wave located in the tip region (Legras et al., 2010).
Figure 8: Analysis of the terms of the steady axial and radial momentum equations for the respective control volumes of the smooth and grooved casings shown in Fig. 7.

Results presented in Fig. 8(a) for the smooth wall configuration highlight the presence of four main net axial forces being applied on the control volume. Two different groups of forces can be distinguished. The first one is the class that acts in opposite direction of the flow ($z < 0$) and counts (i) the net axial force due to the transport of the axial momentum across the radial faces $\sum \Delta (\rho W_z A_r)$ and (ii) the axial pressure force $\sum \Delta (P_s A_z)$ (predictable since the pressure rises across the rotor). The second group includes the forces directed along the flow direction ($z > 0$) that balances the previous group of forces. It counts (a) the net axial force due to the transport of the axial momentum across the longitudinal faces $\sum \Delta (\rho W_z^2 A_z)$. It is also composed of (b) the net axial shear force on the radial faces of the control volumes $\sum \Delta (\tau_{rz} A_r)$ induced by casing and blade tip boundary layers or tip leakage flow. All other terms are zero due to periodicity or to the radial thickness of the control volume. Moreover, Fig. 8(a) clearly shows that fluxes $\sum \Delta (\rho W_z W_r A_r)$ and $\sum \Delta (\rho W_z^2 A_z)$ are both approximately in opposite magnitude (a ratio of 3/4 is observed), thus self-balancing each other. Unlike Shabbir et al. (2005) low speed smooth wall configuration (2005), the blade tip flow of the NASA Rotor 37 is more influenced by the convective forces characterized by higher magnitude of $\sum \Delta (\rho W_z W_r A_r)$. Thereby, the radial velocity $W_r$ is of a primary interest for the equilibrium of the axial momentum equation in the tip flow region. This trend can be explained by (i) the concave shaped endwall of the NASA Rotor 37 (Shabbir et al. test case has a cylindrical shape), which drives the near casing flow radially in the blade passage with high magnitude of $W_r$ and by (ii) the presence of strong velocity gradients (induced by the tip leakage vortex breakdown, by the blade passage shock and the interaction between the leakage flow and the main crossflow (Fig. 4)).

Results in Fig. 8(b) confirms the influence of the velocity component $W_r$ on the radial momentum equation and, moreover, on the radial equilibrium. In fact, it is clearly obvious in Fig. 8(b) that the radial pressure gradient $\sum \Delta (P_s A_r)$ is regulated by the high magnitude of the convective forces $\sum \Delta (\rho W_z W_r A_z)$ and $\sum \Delta (\rho W_r^2 A_r)$. Therefore, the radial equilibrium in the control volume, neglecting the part of the viscous terms, can be simplified as follows:

$$\frac{\partial \rho V_z V_r}{\partial z} + \frac{\partial r \rho V_r^2}{r \partial r} + \frac{\partial P_s}{\partial r} = \frac{\rho V_r^2}{r}$$

(10)

Concerning the grooved casing configuration, results of the axial momentum balance in Fig. 8(a) show that the term of pressure rise across the rotor $\sum \Delta (P_s A_z)$ is maintained, in agreement with the predicted overall performances (Fig. 2(a)). As it has been seen in Fig. 5, there is fluid moving
radially into and out of the slots, which can thus be expected to contribute additional terms to balance the axial and radial momentum equations. Results in Fig. 8(a) reveal the same type of forces acting on the near grooved casing flow than for the smooth one. However, the magnitude of the net axial force $\sum \Delta (\rho W_z W_r A_r)$ has decreased commensurate with the decrease of $\sum \Delta (\rho W_z^2 A_z)$ and viscous force $\sum \Delta (\tau_{rz} A_r)$ (to approximately half the value of the smooth casing configuration). Thereby, the main role of circumferential casing grooves in the current compressor is to turn or drive the force $\sum \Delta (\rho W_z W_r A_r)$ more along the main flow direction i.e. $z > 0$. This observation is in agreement with results of Shabbir and Adamczyk (2005). This is made possible through the change of the radial velocity $W_r$ induced by flow exchange between grooves and blade passage. This observation supports that $W_r$ is quite important for the flow stabilization in the tip region for this concave shaped casing configuration.

Results in Fig. 8(b) show that terms rely to $W_r$ in the radial momentum equation have greatly decreased, in agreement with the results of the axial momentum balance. Regarding the Eq. 10 and assuming that the absolute circumferential velocity $V_{\theta}$ is unchanged compared to the smooth wall configuration, casing grooves reduce the radial pressure gradient $\sum \Delta (P_s A_r)$. This is confirmed in Fig. 8(b). Consequently, the tip flow in presence of grooves is less deviated in a meridian plane.

**Individual Groove Contribution: 1-D Analysis**

This paragraph investigates the axial evolution of the main terms involved in the axial momentum equation (Fig. 9) and the radial momentum equation (Fig. 10). These plots provide information on individual groove contributions.

Results in Fig. 9 and 10 of the smooth wall configuration show that the efforts evolve in amplitude along the axial direction in a nonlinear manner. Moreover, two regions can be clearly distinguished, denoted zone A ($x/C \approx [-20 : 5\%]$) and zone B ($x/C \approx [18 : 45\%]$), which are characterized by a net increase in amplitude of the following convective forces: $\sum \Delta (\rho W_z W_r A_r)$ and $\sum \Delta (\rho W_z^2 A_z)$ in the axial momentum equation (Fig. 9), $\sum \Delta (\rho W_r W_z A_z)$ and $\sum \Delta (\rho W_r^2 A_z)$ in the radial momentum equation (Fig. 10). It is worth noticing that those couples of forces progress along the axial blade chord with the same shape and approximately in opposite magnitude. Moreover, one can observed that the viscous term $\sum \Delta (\tau_{rz} A_r)$ begin to grow at the start of the zone B.

Figure 9: Evaluation of the cumulative sum in the axial direction of the principal terms involved in the axial momentum equation.

Now it is instructive to rely these trends to the flow physics. Fig. 11 shows at mid gap the flow field of balance of the $\sum \Delta (\rho W_z W_r A_r)$ (appearing in the axial momentum equation) with contours
of axial velocity $W_z$. This allows to locate regions of significant force acting along the opposite direction of the flow (i.e. $z < 0$). Results obtained permit to link the growth rate in zone A of the term $\sum \Delta (\rho W_z W_r A_r)$ (i) to the flow acceleration occurring between the detached blade shock and the neighbouring blade passage shock, and (ii) to the flow migration in lower radius due to the concave shaped casing and leading to negative value of $W_r$. Similarly, the second region located in the zone B of rapid growth rate is linked to the tip leakage flow and its abrupt change downstream the blade passage shock. Note that the zone B also corresponds to the beginning of the shear force growth rate ($x/C \approx 20\%$). Results in Fig. 12 confirm that the viscous term $\Delta (\tau z_r A_r)$ is mainly active due to the tip leakage flow and the detached shock from the leading edge.

![Flow field denoted at mid-gap of positive values of $\Delta (\rho W_z W_r A_r)$ with $W_z$ contours.](image1)

Figure 11: Flow field denoted at mid-gap of positive values of $\Delta (\rho W_z W_r A_r)$ with $W_z$ contours.

![Flow field denoted at mid-gap of negative values of $\Delta (\tau z_r A_r)$ with $W_z$ contours.](image2)

Figure 12: Flow field denoted at mid-gap of negative values of $\Delta (\tau z_r A_r)$ with $W_z$ contours.

The effect of circumferential casing grooves is well highlighted in Fig. 8. Grooves influence can be divided into two main regions separated at $x/C = 15\%$. This location corresponds approximately to the axial position of the middle of the 2nd groove and also to the origin of the blockage growth observed in the smooth wall configuration. Upstream $x/C = 15\%$, no difference appears between smooth and grooved casings distributions, thus supporting evidence that the 1st groove does not strongly impact the near casing flow. At $x/C = 15\%$, the 2nd slots significantly impacts the forces
distributions by reducing the growth rates of the following forces: \( \sum \Delta (\rho W_z W_r A_r) \) in axial momentum (Fig. 9) and \( \sum \Delta (\rho W_r^2 A_r) \) in the radial momentum equation (Fig. 10). From \( x/C = 15\% \) to \( x/C = 55\% \), the differences in magnitude of those terms between smooth and grooved casings continue to grow, thus confirming that the 3\( ^{rd} \) and 4\( ^{th} \) grooves are useful. Downstream \( x/C = 55\% \), the curves growth rates are similar to those of the smooth casing configuration, showing that the last groove is useless. These interpretations on grooves usefulness are consistent and in agreement with previous physical analysis of the radial velocity flow field at blade tip (Fig. 5). The difference being that the model can quantify the physical mechanisms induced by the interaction between grooves and the main flow. Furthermore, Fig. 12(b) clearly shows that grooves greatly reduce the net axial force due to the viscous term \( \Delta (\tau_z A_r) \) acting on the near casing flow. However, one can notice that the small “smooth area” between two cavities is even capable of developing viscous efforts.

Compared to the smooth casing configuration, the grooves permit to control the zone B through change of the component \( W_r \) whereas zone A remains uncontrolled (Fig. 11(b)). One reason comes from the axial position of the grooves, which are located further downstream the zone A. Furthermore, the previous consideration of NASA Rotor 37 insensitivity to slot, viewed in literature, is believed to be cause by a lack of control in this zone. A study (not shown) investigates the control of this region by changing the field of \( W_r \) in the vicinity of the zone A. To do so, bleed or injected mass flow at the first groove bottom are tested while keeping the casing treatment geometry (for computational facility reason). Bleeding effect is assumed to smooth the flow deviation occurring in zone A by driving the flow with a more positive value of \( W_r \). In contrast mass flow injection is expected to energize the rotor tip flow so that it can provide force along the blade passage flow direction (i.e. \( z > 0 \)).

CONCLUSIONS

This paper has presented a generalized method of the Shabbir and Adamczyk approach for uncovering the flow mechanisms induced by casing treatment using CFD numerical simulations. Compared to Shabbir’s approach, this method permits to analyze complex flows, like those induced by casing treatments, thanks to a budget analysis of the Navier-Stokes (un)steady momentum equations. In the current paper, only the equilibrium of the steady axial and radial momentum equations are analysed.

The novel method has been used to investigate the tip flow mechanisms on the transonic NASA Rotor 37 with and without a circumferential casing treatment. The analysis of the smooth wall configuration reveals that the equilibriums of the axial and radial momentum equations in the rotor tip region are more dominated by the convective forces than the pressure and viscous forces. This observation is explained by the presence of a strong radial velocity magnitude \( W_r \) caused by the concave shaped casing and the shock system. The presence of circumferential casing grooves in the test case permits to reduce significantly the constraint of the convective forces through a change of the \( W_r \) field. However, the dominance of viscous efforts in the smooth case tends to minimize the efficiency of the grooves (efficiency obtained by reducing the magnitude of the viscous forces). That explains the small differences in overall performances between the two cases. This observation is under validation on other axial compressor casing geometry.

A perspective of this work will be the study of non-axisymmetric casing treatment by taking into account the unsteady set of Navier-Stokes equations. Other points of interest will be the investigation of the inflow condition influence on budget analysis in the rotor tip region in order to uncover the basic mechanisms behind the compressor performance behavior and blockage development. Moreover, note that the methodology can be easily taken for uncovering flow mechanisms and designing a large panel of flow passive control device: film cooling, boundary layer aspiration in turbomachinery field as well as in aircraft or helicopter domain.
ACKNOWLEDGEMENTS

The authors are grateful to Snecma, SAFRAN Group, for permission to publish results. Special thanks to LMFA and CERFACS-CFD Team for their computational facilities as well as for their availability for discussions concerning numerical methods and physical analysis. The authors are also grateful to the elsA software team (ONERA).
REFERENCES


Denton J.D. Lessons from Rotor 37, 3rd ISAIF, Beijing, China, 1996.


Haixin Chen, Song Fu and Huang Xudong CFD Investigation on Tip Leakage Flow and Casing Treatment of a Transonic Compressor, 17th ISABE, paper 1098, 2005.


