# Large-Eddy Simulation of the Flow Developing in Static and Rotating Ribbed Channels

In the present work, the turbulent flow fields in a static and rotating ribbed channel representative of an aeronautical gas turbine are investigated by the means of wall-resolved compressible large-Eddy simulation (LES). This approach has been previously validated in a squared ribbed channel based on an experimental database from the Von Karman Institute (Reynolds and rotation numbers of about 15,000 and  $\pm 0.38$ , respectively). LES results prove to reproduce differences induced by buoyancy in the near rib region and resulting from adiabatic or anisothermal flows under rotation. The model also manages to predict the turbulence increase (decrease) around the rib in destabilizing (stabilizing) rotation of the ribbed channels. On this basis, this paper investigates in more detail the spatial development of the flow along the channel and its potential impact on secondary flow structures. More specifically and for all simulations, results of the adiabatic static case exhibit two contra-rotating structures that are close to the lateral walls of the channel induced by transversal pressure difference created by the ribs. These structures are generated after the first ribs and appear behind all inter-rib sections, their relative position is partly affected by rotation. When considering the stabilizing rotating case, two additional contra-rotating structures also develop along the channel from the entrance close to the low-pressure wall (rib-mounted side). These vortices are due to the confinement of the configuration, inflow profile and are the result of Coriolis forces induced by rotation. Görtler vortices also appear on the pressure wall (opposite to the rib-mounted side). In the destabilizing rotating case, these two types of secondary structures are found to co-exist, and their migration in the channel is significantly different due to the presence of the ribs on the pressure side. Finally, it is shown that heat transfer affects only marginally the static and stabilized cases while it changes more significantly the flow organization in the destabilizing case mainly because of enhanced heat transfer and increased buoyancy force effects. [DOI: 10.1115/1.4046267]

Keywords: computational fluid dynamics (CFD), large-Eddy simulation (LES), heat transfer and film cooling

# Florent Duchaine<sup>1</sup>

CFD Team CERFACS, Toulouse, France e-mail: florent.duchaine@cerfacs.fr

# Laurent Gicquel

CFD Team CERFACS, Toulouse, France e-mail: lgicquel@cerfacs.fr

# Thomas Grosnickel<sup>2</sup>

CFD Team CERFACS, Toulouse, France e-mail: thomas.grosnickel@safrangroup.com

## **Charlie Koupper**

Safran Helicopter Engine, Bordes, France e-mail: charlie.koupper@safrangroup.com

### Introduction

Need for increased efficiency as well as power for gas turbine engines, accompanied by the need for cleaner engines, results in ever-increasing combustor exit temperature with the appearance of hot spots that well surpass existing melting materials currently available for turbine components. To guarantee life span and keep material constraints under acceptable limits, rotating and static parts of the first turbine stage require heavy cooling which can be obtained through various technological choices. In the case of rotating wheels, cooling channels are usually built in the blade solid material so as to extract as much heat as possible from the blade surface to alleviate the heat load resulting from the hot stream in the vein. To enhance heat extraction from the solid to the internal channel flow, while promoting mixing, obstacles are usually added to the channel, the objective is to maximize heat extracting while diminishing as much as possible pressure losses. Although present since a long time in the design of aeronautical engines, this specific problem remains highly misunderstood mainly due to the complex flow physics present in such conditions. Not only heat transfer is present at the interface between the solid part and the internal flow, but also rotation speeds are usually large and

buoyancy, which may appear impacting significant flow responses. Numerical approaches to solve these flows are in industry: Reynolds-averaged Navier–Stokes (RANS), unsteady-RANS [1,2], and more recently point to large-Eddy simulations (LESs) [3–5]. Particular attention is put in the present work on the flow development along the channel with a special focus on secondary flows and the effect of rotation in stabilizing, static, and destabilizing conditions with LES. Contrarily to most literature studies on such flows, real geometries are complex and short which is not fully representative of most works where the flow is assumed to be fully developed flows. One objective is thus to address the spatial development of such flows and more specifically its impact on secondary flow features as they appear through the channel for different operating conditions, i.e., in stabilizing, static, destabilizing, and adiabatic or non-isothermal conditions.

The expected dynamics for a rotating straight channel can be detailed by considering the simple case of a 2D channel flow in the (x, y) plane rotating around the *z*-axis, as shown in Fig. 1.



Fig. 1 Scheme of a 2D-rotating channel

<sup>&</sup>lt;sup>1</sup>Corresponding author.

<sup>&</sup>lt;sup>2</sup>Present address: Safran Helicopter Engines, Bordes, France.

Contributed by the International Gas Turbine Institute (IGTI) of ASME for publication in the JOURNAL OF TURBOMACHINERY. Manuscript received October 30, 2019; final manuscript received January 13, 2020; published online February 6, 2020. Assoc. Editor: David G. Bogard.

Due to rotation, the Coriolis force will balance the wall-normal pressure gradient that will mark the mean flow profile as evidenced in Fig. 1. For this specific flow profile, stability analyses developed by Bradshow et al. [6] underline that boundary layers on the pressure side (bottom part for this specific rotation) will destabilize, while the ones on the suction side will stabilize. The main impact on the flow quantities of such differences in stability is a direct change in the near-wall turbulence levels. More precisely, the stabilized (destabilized) side sees a decrease (increase) in turbulence intensity, implying an increase in symmetry loss of the main flow velocity profile when compared with a laminar rotating channel. Previous observations have been experimentally reported by Johnston et al. [7] and later confirmed by the direct numerical simulation of Kristoffersen and Andersson [8] and Lamballais et al. [9].

Addition of ribs on the bottom side of the above-discussed channel, turbulent flow in the vicinity of the ribs will be either further stabilized or destabilized when the channel is rotating in the negative/positive direction for the reasons detailed above but applied to an already turbulent flow. Rib size relative to the initial smooth channel height will of course be of importance just like the operating flow condition: i.e., base flow Reynolds number and rotational speed. All of these observations were partly confirmed by previous LES studies on such problems postulating either in the experiments or numerically that the flow is fully developed. Such a state is however not guaranteed and may appear only after a specific distance from the inflow condition. This distance of establishment is in itself not fully understood and depends on many parameters including the inflow state. More specifically, for real engine applications, it is often reported that the flow in cooling rotating blade channels never reaches this ideal fully developed state. Therefore, the computational fluid dynamics tool validation needs to highlight such point as discussed hereafter for an experimental benchmark test case from the Von Karman Institute (VKI).

This paper is organized as follows. First, the experimental setup is presented followed by a description of the numerical method used to simulate the test rig. Then, general considerations on the flow development along the ribbed channel are provided, and numerical results are validated through comparisons with the experimental data. Finally, the last section details the sensitivity of the secondary flow structure to the operating point.

#### **Experimental Test Case**

To analyze the flow development in static and rotating heated ribbed channels, the rotating VKI experimental facility by Di

Table 1 Summary of ribbed channel characteristics

Channel length, <i>l</i>	760 mm
Channel height, H	83 mm
Channel depth, w	75 mm
$D_h$	79 mm
Ribs height, h	8 mm
Blockage ratio, $h/D_h$	0.1
Pitch, <i>p/h</i>	10
Ribs angle	90 deg

Sante et al. [10] is investigated. The rig is equipped with an on-board PIV system that allows to measure classical mean velocity quantities from uncorrelated time windows as well as fluctuating quantities resolved in time. The experimental test section is presented in Fig. 2 [11]. It consists of a straight square channel installed on a rotating wooden disc with eight ribs mounted on one side of the channel. The ribs are square with a section of  $h \times h = 8 \times 8 \text{ mm}^2$  and are placed perpendicular to the main flow. The distance separating the ribs is pitch = 80 mm. The geometrical characteristics of the channel are summarized in Table 1. Three cases will be investigated: static channel, positive and negative rotation with respect to the trigonometry direction. The rotation speed retained for this study is around 130 rpm yielding a rotation number Ro = 0.38 defined as

$$Ro = \frac{\Omega D_h}{U_h} \tag{1}$$

where  $\Omega$ ,  $D_h$ , and  $U_b$  are, respectively, rotation speed, channel hydraulic diameter, and bulk velocity of the flow. For these conditions, the Reynolds number based on the bulk velocity and hydraulic diameter is 15,000. Experimentally, the flow comes from the center of the disk through an elbow and is straightened by a honeycomb before entering the channel. Another honeycomb is placed at the outlet of the channel to avoid perturbation of the flow in the test section by ambient flow perturbations when rotating.

Figure 2 illustrates the channel via a side view. The PIV system allows to perform measurement on six windows. Two of them (wa and wb) are located at the inlet of the test section providing information for inlet conditions of numerical simulations. The other windows (w1–w4) are placed between ribs 6 and 7, covering only one-third of the channel height.

#### **Numerical Method**

LES Solver. The LES solver AVBP [12] developed by CERFACS and IFPEN is used to solve the filtered compressible Navier-Stokes (NS) equations. This solver runs efficiently on massively parallel machines and handles unstructured mesh allowing the easiest meshing of complex geometries when compared with structured grids [13]. The convective terms are discretized with an explicit two-step Taylor Galerkin finite element scheme based on a cell vertex formulation [14]. This numerical scheme has good spectral properties (low dissipation and dispersion) and features a third-order accuracy in time and space as required for high-fidelity LES. A second-order Galerkin scheme is used for the diffusion terms [15]. As the temporal integration is explicit, the time-step is limited by the acoustic Courant-Friedrichs-Lewy number which is set to 0.7. The resulting time-step controlled by the size of the smallest cells located close to ribbed walls is of the order of  $dt \approx 0.6 \,\mu s$ . The sub-grid scale (SGS) viscosity  $\nu_t$  is obtained by the wall adapting local Eddy-viscosity model [16]. This model is well suited for wall-resolved wall-bounded flows as it is constructed to recover the scaling laws of turbulent viscosity at walls [16]. A classical gradient-diffusion hypothesis is used to compute SGS heat fluxes [17]. SGS heat fluxes are computed with the filtered temperature gradient using an SGS thermal



Fig. 2 Experimental test bench: schematic lateral view of the PIV zones not at scale

Fig. 3 Scheme of the numerical domain

conductivity  $\lambda_t$ , which is linked to the SGS viscosity  $\nu_t$  through an SGS turbulent Prandtl number here fixed at  $Pr_t = 0.6$ .

**Boundary Conditions and Mesh.** The numerical domain retained for the present study is presented in Fig. 3. Velocity profiles measured experimentally for the different cases are imposed at the inlet along with a uniform temperature,  $T_0 = 293$  K, using the Navier–Stokes characteristic boundary condition (NSCBC) formalism [18]. No turbulent fluctuations are added since the experimental inlet has been designed to obtain a symmetric inlet velocity profile, free of secondary flows, and with a turbulent intensity lower than 1% [5]. At the outlet, static pressure is enforced using the NSCBC formalism accounting for the transverse terms [19]. All walls (S, T, and R of Fig. 3) are adiabatic, and no-slip for the reference adiabatic cases. Dealing with anisothermal cases, the flow is heated by the bottom wall (R) using a Neumann iso-flux and no-slip boundary condition.

To accurately resolve the aerodynamic and thermal boundary layer in a wall-resolved framework at minimal cost when compared with a full tetrahedral mesh, a hybrid tetrahedra/prismatic grid approach is adopted. Indeed, for the same spatial resolution in the normal direction, a prism layer uses less elements and leads to a higher minimum cell volume than a full tetrahedral-grid approach as prismatic elements can have a large aspect ratio. All the walls are meshed with a layer of prismatic elements where the height of the prisms is smaller than the size of their triangle basis by a ratio of about 5 to control numerical errors. The mesh used for the static and negative rotation cases has 3.6 million cells with a mean  $y^+$  on the ribbed wall (R) of about 3 without exceeding 5 as required for a wall-resolved LES (Fig. 4). The characteristics of this mesh are the result of a convergence study performed by Fransen et al. [2] and used by Scholl et al. [20] as well as Grosnickel et al. [21]. Concerning the positive rotation, the destabilization of the flow due to rotation takes place close to the ribbed wall. As a consequence, a finer mesh of 9.5 million cells is used as described



Fig. 4 Distribution of  $y^+$  on the ribbed wall for the (a) positive rotation, (b) static, and (c) negative rotation cases in the adiabatic condition (gray on the bottom ribbed wall and black on the ribs)



Fig. 5 Q-criterion colored by the normalized axial velocity in the domain and vorticity magnitude in the symmetry plane based on an instantaneous solution of the VKI channel configuration in the static condition

in Grosnickel et al. [21]. The mean  $y^+$  for this refined mesh is also about 3 without exceeding 9 on the ribbed wall (Fig. 4).

The rotation of the domain for the positive and negative cases is performed by moving the mesh in a solid rotation using an arbitrary Lagrangian-Eulerian framework [22,23]. This approach removes the need for correcting the Navier–Stokes equations since rotation is transmitted implicitly to the flow-through moving boundaries and mesh points as commonly done in other studies [24–28].

#### **General Considerations on Flow Development**

This section presents general considerations about flow development along the channel in static and rotating cases for adiabatic and then anisothermal conditions. As illustrated in Fig. 5, the flow at the inlet of the channel is smooth and quasi-laminar. Shear layers appear after the first rib before to be destabilized by the following ribs leading to a highly turbulent flow. The time-averaged flow topology is thus directly impacted by the predicted flow development and the ability of the solver to well reproduce the relevant features in space is of primary importance. Although the study of turbulent structures organization is of interest in this configuration, the analyses proposed in this paper focus on the mean topology of the flow only. To do so, time-averaged field obtained on 3 flowthrough times of the whole configuration corresponding to about 30 characteristic convection time between two successive ribs is investigated.

Figure 6 illustrates the spatial development of the flow through the time-averaged axial velocity field in the symmetry plane of the configuration for the adiabatic test cases in static, negative,

and positive rotations. In these three cases, the global behavior of the flow is similar: ribs induce a blockage resulting in a deviation of the flow toward the opposite wall with an elevation in the velocity. The flow topology in the region close to the ribs also presents similar patterns among cases. Indeed, several recirculation zones establish downstream, upstream, and above the ribs and will more deeply investigated later. It is worth noting that in the three operating conditions, the flow appears to be almost established and fully developed after the first three ribs. Indeed, in each case, the recirculation zones located in-between the three first ribs are larger than further downstream where this region of the flow appears to be quasi-periodic along the streamwise direction. When comparing the operating conditions, the main difference between the static and rotating cases lies first in the size of the main recirculation zone downstream of the ribs after the development within the channel. The size of this zone is directly linked to the turbulence levels in the channel and thus to the stabilizing and destabilizing induced by the rotation. The stabilizing and destabilizing effects of the negative and positive rotations are, respectively, confirmed by Fig. 7 where the fluctuations of the axial velocity are displayed through the axial velocity root-mean-squares (RMSs) in the symmetry plane of the channel. Due to the transition into a turbulent flow evidenced in Fig. 5 for the static case, axial velocity RMS starts only to increase around the second rib. The primary source of this turbulence generation is the apparition of the rib-induced shear layers which become unsteady resulting in high RMS spots for the static condition. The negative rotation is here observed to stabilize the shear layers resulting in lower levels of RMS all along the channel when compared with the static case. On the contrary, the destabilizing effect of the positive rotation enhances the axial



Fig. 6 Time-averaged normalized axial velocity on the symmetry plane for the (a) positive rotation, (b) static, and (c) negative rotation cases in the adiabatic condition



Fig. 7 RMS of the axial velocity on the symmetry plane for the (a) positive rotation, (b) static, and (c) negative rotation cases in the adiabatic condition

velocity fluctuations directly after the first rib and leads globally to higher RMS levels in the entire channel. As a result of the turbulent activity, the main recirculation zone has a medium size in the static case taking about half the distance between two consecutive ribs. The stabilizing effect of the negative rotation induces more stable shear layer above the ribs, and the main recirculation zones are larger than that in the static case and occupy the whole inter-rib space. On the contrary, the destabilization induced by the positive rotation leads to high turbulence activity and shorter main recirculation zones. An other important difference between the three operating cases concerns the flow acceleration directly above the rib region. While the axial velocity of the static and negative rotating cases features a homogeneous behavior in the axial direction, it is more erratic in the positive rotating case with two main characteristics. The first one is an acceleration of the flow right above each rib pushing the fluid toward the center of the channel. The second point concerns the low-velocity patterns close to the top smooth wall which are due to secondary flows developing along the channel as discussed later in the paper. These secondary flow structures are weakly unsteady as evidenced by the axial velocity fluctuations that are generated close to the smooth top wall in the upper part of the channel.

The impact of heating the wall equipped with ribs on the timeaveraged axial velocity development is illustrated in Fig. 8 for the three cases. Concerning the static case, the aerodynamics of the anisothermal simulation is very similar to the adiabatic one. In the positive rotation case, heat transfer has an important impact on the flow development along the channel both in the rib region and the main passage. First, wall heat addition induces larger mean recirculation zones mainly in the upstream part of the channel. Then, the secondary flow structures evidenced in the adiabatic case on the smooth wall have a weaker impact on the flow and are pushed closer to the wall certainly due to the different flow organization of recirculation zones in the inter-rib regions creating a higher blockage. This difference implies a migration of the high-speed zone evidenced by  $U/U_b > 1.2$  above ribs 3 to 8. For the negative rotation case, major differences appear in the inter-rib regions. When the flow is heated by the ribbed wall, the axial velocity above y/h = 1 (i.e., in the clean stream) is much smaller than with the adiabatic wall condition. The differences are more pronounced after the third rib. The main recirculation is lager and thus takes more space between the ribs in the anisothermal condition. The main difference is the elevation in the high-velocity zone of  $U/U_b > 1.2$  toward the smooth top wall of the channel leading to a decrease in the axial velocity close to the ribs. Coletti et al. [29] have already reported experimentally this phenomenon linked to centripetal buoyancy force acting on the heated fluid. These consequences are large recirculations of hot gases taking place in whole inter-rib regions as well as a displacement of the main stream away from the ribs by the mixing layer as evidenced in Fig. 8. It was shown numerically by Sewall et al. [28] that for a Buoyancy number Bo lower than 0.25, the recirculation zone is growing



Fig. 8 Time-averaged normalized axial velocity on the symmetry plane for the (a) positive rotation, (b) static case, and (c) negative rotation in the non-isothermal condition

when increasing Bo, and for Bo > 0.25, the recirculation takes all the inter-rib space, with Bo defined as:

$$Bo(r) = r\Omega^2 \frac{D_h}{U_0^2} \frac{T_w - T_0}{T_0}$$
(2)

where *r* is the local radius,  $r\Omega^2$  is the centrifugal force, and  $T_w$  is the wall temperature, while  $U_0$  and  $T_0$  are, respectively, the bulk velocity and bulk temperature. Buoyancy is mainly driven not only by the temperature difference issued by the wall and the fluid but also by the rotation number. In the present case, Bo = 0.2 corresponds to the third rib location where the flow topology starts to change illustrating a strong coupling between aerodynamics and heat transfer in the studied anisothermal operating condition.

#### Validations of Large-Eddy Simulation Results

The validation of the different LES results is obtained by comparing time-averaged axial velocity fields from the computations with experimental data in the region between ribs 6 and 7 as shown in Fig. 2. The recirculation zones between these two ribs for the different operating points (adiabatic/heat flux, static/rotating cases) are visualized by the use of restricted 2D streamlines in the symmetry plane and are represented in Fig. 9. The discontinuities in the experimental fields are due to the different PIV acquisition windows illustrated in Fig. 2. As mentioned, the thermal condition does not affect the aerodynamics of the static case so only the adiabatic case is represented. For this specific case, the LES well reproduces the size of the four recirculation zones : main recirculation downstream of rib 6 which rotates clockwise, the induced bubble close to rib 6 foot rotating anti-clockwise, and the recirculation zones before and on top of rib 7 that rotates clockwise. The main modifications of these recirculation zones induced by positive and negative rotations are well captured by the simulations: a reduction of the principal recirculation zone for the positive destabilizing rotation and an increase in the case of the negative stabilizing rotation. Simulation and experimental results compare also well when heat transfer is added: the enlargement of the main recirculation zone is well predicted by LES in both rotating cases.

Figure 10 shows more quantitative comparisons by directly comparing profiles at various axial positions in the inter-rib section as obtained by the computations and measurements for the adiabatic condition. This further confirms the ability of LES to reproduce the experimental axial velocity in the static and rotating cases. The intensity of the main recirculation zone and the velocity levels reached outside the rib region (around y/h = 2) is also well predicted for all three operating conditions. The velocity fluctuations are also well predicted for the three cases as illustrated by



Fig. 9 Streamlines based on time-averaged velocity between ribs 6 and 7 for the (a) positive rotation, (b) static, and (c) negative rotation under adiabatic and anisothermal conditions



Fig. 10 Time-averaged axial velocity profiles between ribs 6 and 7 from adiabatic LES (lines) and PIV (symbols): (a) positive rotation, (b) static, and (c) negative rotation

Fig. 11 where the axial velocity RMS profiles obtained by LES are compared with experimental measurements. The topology of the profiles is well captured by the simulations with the peak of activity in the shear layer induced by the separation behind the rib. A slight overestimation of the RMS levels at this peak is observed at some axial locations for the positive and static cases. On the contrary, the negative case LES result exhibits a global underestimation of the RMS when compared with measurements. It seems that the numerical method (mesh, numerical scheme as well as SGS model) leads to a more pronounced stabilization for the negative rotation than in the experiment.

Figure 12 shows the impact of imposing an isothermal boundary condition on the ribbed wall for the static and the two rotating cases in the region of interest that is between ribs 6 and 7. As anticipated from the previous section, when the channel is rotating, bigger differences appear between adiabatic condition and isothermal one than in the static case. For the positive rotation case, the major modification concerns the increase in intensity and size of the main recirculation zone after rib 6. When the wall is heated, the recirculation becomes stronger with a more important negative axial velocity when y/h < 1. The size of the recirculation zone between the two ribs is also larger: reattachment occurs between x/h = 4 and x/kh=5 for the adiabatic case while the flow is still recirculating at x/h = 9 for the anisothermal case. Finally, no important difference exists close to the channel's center for y/h above 1.5. For the negative rotation case, the main recirculation zone is slightly more important. The main difference in this case lies in the level of axial velocity in the region just above the ribs (1 < y/h < 3) which is lower in the anisothermal case due to an increase of the blockage induced by buoyancy and affecting the inter-rib region. The LES/ experiment comparison is good, especially in the vicinity of the ribs even if LES predicts a junction of the adiabatic and anisothermal profile for higher values of y/h than the experimental results.

To characterize wall heat transfer, the enhancement factor (EF) that compares the local Nusselt number Nu to a reference Nusselt

number Nu<sub>0</sub> is used:

$$EF = \frac{Nu}{Nu_0}$$
(3)

In this work, the reference Nusselt number is obtained with the Dittus–Boelter relation  $Nu_0 = 0.023 \text{Re}^{0.8} \text{Pr}^{0.4}$ . Several possibilities exist to compute the Nusselt number Nu. The methodology to obtain Nu with LES data which slightly differs from the experimental protocol [5] is exposed hereafter. The Nusselt number is numerically computed by

$$Nu = \frac{h_c D_h}{\lambda} \tag{4}$$

where  $\lambda$  is the thermal conductivity at the wall temperature. The heat transfer coefficient  $h_c$  is determined following the regression method proposed by Fenot et al. [30] using four LES for each rotating condition with four different wall heat fluxes  $q_w = 500$ , 1000, 1500, and 2000 W/m<sup>2</sup>. As the determination of the Nusselt number differs between the experimental and numerical protocols, only the shape of the EF on the wall between ribs 6 and 7 presented in Fig. 13 can be compared. The global tendency of EF augmentation for a positive rotation due to flow destabilization as well as EF reduction for the negative rotation due to flow stabilization and larger recirculation induced by buoyancy forces discussed before is well captured by the computations. Then, the general patterns are well predicted by the LES. The first important aspect is the longitudinal organization with a low heat transfer close to rib 6  $(0 \le x/h \le 1)$  due to the induced recirculation bubble visible for each operating point. Second, every case exhibits lateral patterns due to secondary flows that will be discussed in more detail in the section dedicated to secondary flow analysis. The LES reproduce the EF increase in the lateral corners for negative as well as positive rotation for which this effect is more pronounced.



Fig. 11 RMS of axial velocity between ribs 6 and 7 from adiabatic LES (lines) and PIV (symbols): (a) positive rotation, (b) static, and (c) negative rotation

To conclude this section, the unsteady simulations performed with LES are able to reproduce with a high fidelity the mean axial velocity measured experimentally in the region between ribs 6 and 7 for the different operating points: static and rotating as well as adiabatic and with heat transfer applied to the ribbed wall. The same conclusion can be drawn on transverse velocity as well as on RMS. These LES/experiment agreements confirm that the computations also well capture the development of the flow topology along the channel and that such effects need to be carefully captured for specific these conditions since no true fully developed flow is effectively obtained here. This observation was found to be especially critical in the destabilizing adiabatic condition. As shown previously, the flow development is highly dependent on the operating condition and a fully periodic behavior is not reached at the end of the channel mainly for the negative rotation case and potentially but to a lesser extent for other conditions. For the negative rotation, secondary flows on the smooth wall opposite to rib wall are observed to play an important role in the velocity distribution within the channel and therefore in the inter-rib spacing as the flow develops. Such a feature was also found to depend on the inflow profile [21]. The next section discusses in more detail the secondary flow structures for this configuration as well as their sensitivities to rotation and wall heat flux.

#### **Secondary Flow Structures**

Comparisons of the temporally averaged mean axial velocity maps in the symmetry plane of the configuration confirm that flow establishment within the channel from the inlet section to its fully developed state is a strong function of the operating condition. One reason for such behaviors relates to the flow organization and flow response to the rib-induced blockage and more specifically to the generation of secondary flows that can be prompted by rotation. The result is different competitive forces acting on the main axial stream and resulting in different transversal flow organizations. These features are first studied for the adiabatic cases in Fig. 14 by focusing on four transversal planes located through the channel and positioned, respectively, at (a), (e), and (j): middistance between the inlet and the first rib; (b),(f), and (j): middistance between the second and third rib; (c), (g), and (k): middistance between the fourth and fifth rib; and (d), (h), and (l): middistance between the sixth and seventh rib. All frames show the non-dimensional mean axial velocity field within the transversal section of the channel with the addition of streamlines restrained to this plane to evidence flow recirculations and overall organization in each transversal plane.

Focusing first on the reference case that is the static adiabatic channel, Figs. 14(e)-14(h), distinct and well-identified secondary flow structures establish as it proceeds through the channel. Initially unperturbed and solely axial in the first section, Fig. 14(e), the flow is rapidly affected by the presence of the ribs on the bottom section by generating two contra-rotating structures induced by the conjunction of the lateral walls and the rib-induced pressure gradient. As a result and at mid-distance in the inter-rib passage, two slowly rotating structures located at mid-height in the passage redistribute the flow extracting mass from the inter-rib region along the lateral walls and injecting mass from the main channel stream in the inter-rib region in the symmetry plane of the configuration. Additionally to these two main features, four contra-rotating appear on the bottom wall in Fig. 14(f) whose height appears to scale with the rib height and seem to remain in latter sections but appear to be much weaker. Focusing on the stabilizing case, Figs. 14(i)-14(l), a clear impact of rotation is evidenced. Although stabilizing, secondary structures are more diverse and appear. Dissymmetry in the first section, Fig. 14(i), is explained through the inflow condition



Fig. 12 Time-averaged axial velocity profiles between ribs 6 and 7 from adiabatic and anisothermal LES (lines) and PIV (symbols): (a) positive rotation, (b) static, and (c) negative rotation



Fig. 13 Enhancement factor contours obtained experimentally (up) and numerically (down) for the rotating and static cases between ribs 6 and 7



Fig. 14 Comparison of secondary flow structures obtained with adiabatic simulations for positive rotation, static, and negative rotations at (a), (e), (j): mid-distance between the inlet and the first rib; (b), (f), (j): mid-distance between the second and third rib; (c), (g), (k): mid-distance between the fourth and fifth rib; and (d), (h), (l): mid-distance between the sixth and seventh rib

that is specified in agreement with measurements and Fig. 1. The consequence of this inflow modification and presence of the ribs is the separation of the channel cross section into two distinct flow region: (a) the bottom half of the channel including the interrib region produces an overall flow organization similar to the static channel case but in half the spatial extent and (b) the top part of the channel or pressure side in this specific case, which produces four vortices assimilated to Taylor-Görtler [31] structures. These later clearly appear as going through the channel, gaining in strength to a point where they potentially interact and migrate within the top section of the channel. One other important feature that distinguishes this flow from the static case is the apparent confinement to the lateral walls of two rotation-induced contra-rotating vortices. These are present directly after the entree section migrates slowly above the ribs and accentuates the previously identified downward flow observed in the static case. This case distinguishes itself from the previous one by the fact that the downward flow starts from the mid-section flow to the inter-rib region when it appears to come from the top section of the channel flow in the static case. It furthermore covers a smaller transversal extent in the static case. Likewise, the dynamics in the inter-rib region appears to be stronger with the existence of four sustained vortices directly behind the ribs through the entire length of the channel. Switching to the destabilizing case, Figs. 14(a)-14(d), conclusions again differ. Similarly to the stabilizing case and in agreement with the prescribed inflow condition, the initial section exhibits a clear dissymmetry, opposite to the stabilizing case. Again, rotation-induced structures are seen here to appear directly after the channel entree and position in the top section, the suction side of the channel, growing in strength as the flow advances through the channel. Similarly to all cases, four contra-rotating vortices appear behind the ribs and are present in all sections. The main stream and top streams are however significantly affected in this case. First, the rotation-induced structures rapidly expand through the top part of the channel section, Fig. 14(b), interacting with the two rib-induced structures. This interaction seems to evolve further downstream producing different patterns of the mean axial velocity fields for the different downstream sections confirming a non-fully established flow even near the exit of the channel. Second, these interactions are observed to impact significantly the organization of the inter-rib flow even if the previously identified rib-induced four structures remain present. Similar flow organizations have been already reported by

Borello et al. [32] as well as Mayo et al. [33] in the context of configurations with an axial periodicity. The authors mentioned the presence of rib-induced vortices as well as Coriolis-driven secondary flows whose positions and intensities defer depending on positive or negative rotation. Taylor–Görtler vortices are also evidenced in the negative rotation case. A major difference is linked to the secondary structures in the inter-rib region mainly visible in the rotating cases not reported by Borello et al. [32] (but visible in the positive rotating case) and Mayo et al. [33].

As discussed previously, the addition of heat transfer to the problem will affect the flow predictions. One primary consequence of heat transfer is the introduction of buoyancy forces which will interact with rotation induced forces and alter the various equilibria by favoring or overshadowing secondary flow structure generations. Interestingly, this is a strongly coupled process as secondary structures play an important role in heat transfer [34]. As a complement to the previously given results and analyses, a comparison between rotating cases in adiabatic and anisothermal conditions is proposed in Fig. 15. Only the cross-stream flow organization is of interest here and is analyzed through the use of two transversal planes, respectively, located (*a*) and (*b*), at mid-distance between ribs 2 and 3, and (*c*) and (*d*), at mid-distance between ribs 6 and 7. Positive and negative rotating cases are, respectively, provided in Figs. 15(*a*), 15(*c*) and 15(*b*), 15(*d*). Finally and to ease the direct comparisons of all flow predictions, the channel section is divided into two sections (under the symmetry hypothesis), the



Fig. 15 Comparison of secondary flow structures obtained with adiabatic and anisothermal simulations for positive and negative rotations at mid-distance between (a), (b) ribs 2 and 3 and (c), (d) ribs 6 and 7

right side corresponding to the adiabatic case while the left side corresponds to the associated anisothermal case. All fields are nondimensionalized by the flow bulk velocity and show the mean axial velocity field complemented by 2D streamlines restricted to the plane of observation. Such views confirm the non-negligible influence on the flow organization of heat addition. For the stabilizing cases, Figs. 15(b) and 15(d), the main impact at both section does not appear, and all secondary flow structures evidenced by the adiabatic case are present with heat addition. Variations in size and strength do not clearly appear, and the only obvious effect is a slight modification of the low axial velocity region linked to the presence of the ribs which shifts slightly upward. For the destabilizing cases, Figs. 15(a) and 15(c), differences are more pronounced especially in the inter-rib region. Indeed in the anisothermal case, heat addition to the flow through the bottom wall favors the growth and extent of two of the previously identified four rib-induced bottom vortices. In fact in the last sections, these vortices grow to a point where they interact with the mid-section flow constraining the initially observed top rotation induced secondary flow structures as well as the two mid-channel rib-induced rotating structures. All of these analyses confirm that with heat addition, once buoyancy is activated: i.e., Bo > 0.2 or after rib number 3, large modifications of the flow fields are expected especially in the destabilizing case and in favor of emerging secondary flow structures over others when compared with the adiabatic cases.

#### **Conclusions and Outlook**

Introduction of cooling systems in the rotating parts of the turbine stage of aeronautical engines will appear mandatory especially in the next generation of engines where performance gains and pollutant emissions constraints will result in increased temperature levels in this part of the engine. Their design is however still a challenge and many multi-physics features are at play. In the case of the ribbed channel, the interaction of the main stream flow with the ribbed surface under rotation is still not clearly understood. Many geometrical factors are indeed at play, rotation being a key player in destabilizing or stabilizing the flow. Rotation-induced secondary flows are present, and their interaction with the ribs differ depending on the rotation direction and addition of heat. The main consequence is that in real engine geometries, the resulting flow is most likely not fully developed. In an attempt to evidence and understand such processes, the entire axial extent of the VKI rotating ribbed channel facility is here simulated by the use of the wallresolved LES under various conditions. As discussed in the present study, all predictions agree with experimental measurements demonstrating the adequacy of the approach to address this configuration. However depending on the operating condition, flow development is clearly observed to differ and to some extent, none of the conditions yield a fully developed flow. One reason for such differences and observations is the establishment of different sec-

#### Acknowledgment

This work was granted access to the HPC resources of CINES under the allocation 2018-A0042A06074 made by GENCI.

#### Nomenclature

- h = ribs height
- l = channel length
- p = pitch
- r = radius

- H = channel height
- T = temperatureU = velocity
- $h_c$  = heat transfer coefficient
- $q_w$  = wall heat flux
- $D_h$  = hydraulic diameter
- $T_0$  = inlet temperature
- $T_w$  = wall temperature
- $U_b$  = bulk velocity
- $y^+$  = normalized wall distance
- Bo = buoyancy number
- Nu = Nusselt number
- Ro = rotation number
- $Nu_0$  = reference Nusselt number
- $Pr_t = turbulent Prandtl number$

#### Symbols

- $\lambda$  = thermal conductivity
- $\lambda_t$  = subgrid-scale conductivity
- $\nu_t$  = subgrid-scale viscosity
- $\Omega$  = rotation speed

#### References

- [1] Liu, Y., Tucker, P. G., and Lo Iacono, G., 2006, "Comparison of Zonal RANS and LES for a Non-Isothermal Ribbed Channel Flow," Int. J. Heat Fluid Flow, 27(3), pp. 391-401.
- [2] Fransen, R., Gourdain, N., and Gicquel, L. Y. M., 2012, "Steady and Unsteady Modeling for Heat Transfer Predictions of High Pressure Turbine Blade Internal Cooling," ASME Turbo Expo 2012: Turbine Technical Conference and Exposition, Paper No. GT2012-69482.
- [3] Sewall, E. A., and Tafti, D. K., 2004, "Large Eddy Simulation of the Developing Region of a Rotating Ribbed Internal Turbine Blade Cooling Channel," ASME Turbo Expo, Vienna, Austria, June 14-17.
- [4] Fransen, R., Vial, L., and Gicquel, L. Y. M., 2013, "Large Eddy Simulation of Rotating Ribbed Channel," ASME Turbo Expo 2013: Turbine Technical Conference and Exposition, Paper No. GT2013-95076.
- [5] Mayo, I., 2017, "Flow Field and Heat Transfer in a Rotating Rib-Roughened Cooling Passage," Ph.D. thesis, Dynamique des fluides, Institut National Polytechnique de Toulouse.
- [6] Bradshaw, P., 1969, "The Analogy Between Streamline Curvature and Buoyancy in Turbulent Shear Flow," J. Fluid Mech., 36(1), p. 177.
- [7] Johnston, J. P., Halleen, R. M., and Lezius, D. K., 1972, "Effects of Spanwise Rotation on the Structure of Two-Dimensional Fully Developed Turbulent Channel Flow," J. Fluid Mech., 56(3), pp. 533–557.
- [8] Kristoffersen, R., and Andersson, H., 1993, "Direct Simulations of Low-Reynolds-Number Turbulent Flow in a Rotating Channel," J. Fluid Mech., 256, pp. 163-197.
- [9] Lamballais, E., Lesieur, M., and Métais, O., 1996, "Effects of Spanwise Rotation on the Vorticity Stretching in Transitional and Turbulent Channel Flow," Int. J. Heat Fluid Flow, 17(3), pp. 324-332.
- [10] Di Sante, A., Theunissen, R., and Van Den Braembussche, R. A., 2008, "A New Facility for Time-Resolved PIV Measurements in Rotating Channels," Exp. Fluids, 44(2), pp. 179-188.
- [11] Coletti, F., Maurer, T., Arts, T., and Di Sante, A., 2012, "Flow Field Investigation in Rotating Rib-Roughened Channel by Means of Particle Image Velocimetry," Exp. Fluids, 52(4), pp. 1043-1061.
- [12] Schønfeld, T., and Rudgyard, M., 1999, "Steady and Unsteady Flows Simulations Using the Hybrid Flow Solver Avbp," AIAA J., 37(11), pp. 1378-1385.
- [13] Gicquel, L., Gourdain, N., Boussuge, J.-F., Deniau, H., Staffelbach, G., Wolf, P., and Poinsot, T., 2011, "High Performance Parallel Computing of Flows in Complex Geometries," Comptes Rendus Mécanique, 339(2-3), pp. 104-124.
- [14] Colin, O., and Rudgyard, M., 2000, "Development of High-Order Taylor-Galerkin Schemes for Unsteady Calculations," J. Comput. Phys., 162(2), pp. 338-371.
- [15] Donea, J., and Huerta, A., 2003, Finite Element Methods for Flow Problems, John Wiley & Sons Inc, New York.
- [16] Nicoud, F., and Ducros, F., 1999, "Subgrid-Scale Stress Modelling Based on the Square of the Velocity Gradient," Flow Turb. Combust., 62(3), pp. 183-200.
- [17] Pope, S. B., 2000, Turbulent Flows, Cambridge University Press, Cornell University, New York.
- [18] Poinsot, T., and Lele, S., 1992, "Boundary Conditions for Direct Simulations of Compressible Viscous Flows," J. Comput. Phys., 101(1), pp. 104-129.
- [19] Granet, V., Vermorel, O., Leonard, T., Gicquel, L., and Poinsot, T., 2010, "Comparison of Nonreflecting Outlet Boundary Conditions for Compressible Solvers on Unstructured Grids," AIAA J., **48**(10), pp. 2348–2364.
- [20] Scholl, S., Verstraete, T., Duchaine, F., and Gicquel, L., 2016, "Conjugate Heat Transfer of a Rib-Roughened Internal Turbine Blade Cooling Channel Using Large Eddy Simulation," Int. J. Heat Fluid Flow, 61(B), pp. 650-664.
- [21] Grosnickel, T., Duchaine, F., Gicquel, L., and Koupper, C., 2017, "Large-Eddy Simulations of Static and Rotating Ribbed Channels in Adiabatic and

ondary flow structures of various strength and initial localization as the flow advances through the channel. Such an organization is furthermore modified in anisothermal cases underlying the complexity of such flows and the difficulty to obtain efficient cooling designs. In that respect, the use of LES may contribute to better the complex underlying flow physics as demonstrated in recent works although more validation seems necessary.

Isothermal Conditions," Proceedings of ASME Turbo Expo 2017: Turbine Technical Conference and Exposition, Charlotte, NC, June 26–30.

- [22] Hirt, C. W., Amsden, A. A., and Cook, J. L., 1974, "An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speeds," J. Comput. Phys., 14(3), pp. 227–253.
- [23] Moureau, V., Lartigue, G., Sommerer, Y., Angelberger, C., Colin, O., and Poinsot, T., 2005, "Numerical Methods for Unsteady Compressible Multi-Component Reacting Flows on Fixed and Moving Grids," J. Comput. Phys., 202(2), pp. 710–736.
- [24] Murata, A., and Mochizuki, S., 2000, "Large Eddy Simulation With a Dynamic Subgrid-Scale Model of Turbulent Heat Transfer in An Orthogonally Rotating Rectangular Duct With Transverse Rib Turbulators," Int. J. Heat Mass Transfer, 43(7), pp. 1243–1259.
- [25] Qin, Z., and Pletcher, R. H., 2006, "Large Eddy Simulation of Turbulent Heat Transfer in a Rotating Square Duct," Int. J. Heat Fluid Flow, 27(3), pp. 371–390.
- [26] Ahn, J., Choi, H., and Lee, J. S., 2007, "Large Eddy Simulation of Flow and Heat Transfer in a Rotating Ribbed Channel," Int. J. Heat Mass Transfer, 50(25–26), pp. 4937–4947.
- [27] Elyyan, M. A., and Tafti, D. K., 2010, "Investigation of Coriolis Forces Effect of Flow Structure and Heat Transfer Distribution in a Rotating Dimpled Channel," ASME Conf. Proc., 2010(43994), pp. 245–254.

- [28] Sewall, E. A., and Tafti, D. K., 2008, "Large Eddy Simulation of Flow and Heat Transfer in the Developing Flow Region of a Rotating Gas Turbine Blade Internal Cooling Duct With Coriolis and Buoyancy Forces," ASME J. Turbomach., 130(1), p. 011005.
- [29] Coletti, F., Jacono, D. L., Cresci, I., and Arts, T., 2014, "Turbulent Flow in Rib-Roughened Channel Under the Effect of Coriolis and Rotational Buoyancy Forces," Phys. Fluids, 26(4), p. 045111.
- [30] Fenot, M., Dorignac, E., and Vullierme, J. J., 2008, "An Experimental Study on Hot Round Jets Impinging a Concave Surface," Int. J. Heat Fluid Flow, 29(4), pp. 945–956.
- [31] Saric, W. S., 1994, "Görtler Vortices," Annu. Rev. Fluid Mech., 26, pp. 379–409.
- [32] Borello, D., Salvagni, A., and Hanjalic, K., 2015, "Effects of Rotation on Flow in An Asymmetric Rib-Roughened Duct: LES Study," Int. J. Heat Fluid Flow, 55, pp. 104–119.
- [33] Mayo, I., Arts, T., and Gicquel, L., 2018, "The Three-Dimensional Flow Field and Heat Transfer in a Rib-Roughened Channel At Large Rotation Numbers," Int. J. Heat Mass Transfer, 123, pp. 848–866.
- [34] Salvagni, A., Borello, D., Rispoli, F., and Hanjalic, K., 2017, "Large-Eddy Simulations of Heat Transfer in Asymmetric Rib-Roughened Ducts: Effects of Rotation," Int. J. Heat Fluid Flow, 68, pp. 373–385.