An experience in eddy resolving simulation of mixed convection in a rotating annular cavity with one heated disk and axial throughflow

A G Abramov, D K Zaitsev, E M Smirnov and A A Smirnovsky

Peter the Great St.Petersburg Polytechnic University, Polytechnicheskaya str. 29, St.Petersburg, 195251 Russia

E-mail: smirnov_em@sbpstu.ru

Abstract. Results of numerical simulation of the flow and heat transfer in a rotating annular cavity with one heated disk and axial throughflow are presented. To assess the model effect on the solution, various vortex-resolving techniques and different computational grids have been applied. The simulation results are compared with experimental data available. Consistent with the measurements, the numerical flow analysis has revealed the presence of two pairs of cyclonic and anti-cyclonic global circulations. It has been found that all computational models tested underestimate considerably both the mean heat flux and the mean rotational velocity lag whereas the results obtained using different models are relatively close to each other.

1. Introduction

Investigation of local heat transfer in rotating annular cavities with heated disks and axial throughflow is of great interest and importance in turbomachinery applications where configurations of that kind are widely used to cool the rotor of axial compressors and gas turbine engines [1]. The rapid rotation and opacity of the cavity walls create great difficulties in obtaining accurate quantitative data on unsteady flow and local heat transfer. Usually, experimental studies are limited to visual observation of the flow structure and measurement of the time-averaged local and integral heat transfer. Among known experimental works on the topic, one can notice papers [2, 3] where systematic data on convection in rapidly rotating annular cavities with and without axial throughflow are presented. These data obtained under well-defined conditions in relatively simple geometry were extensively used for primary validation of three-dimensional computational models [4–6].

For cases of practical interest that are characterized by relatively high Grashof numbers, published CFD studies usually showed poor predictions of local heat transfer on disk surfaces, probably due to insufficient grid resolution. As well, most of the former calculations were carried out on the basis of RANS approach (e.g. [7, 8]) that doesn’t take into account fundamental features of such flows, in particular, the presence of quasi-laminar Ekman layers that play a decisive role in the heat and mass transfer processes near the disks. Currently, it is believed that a deep understanding of the unsteady dynamics and heat transfer in rapidly rotating cavities can only be achieved using LES-based methods that need less empirism for adequate modelling of the turbulent mixing. The advantage of the LES method over the RANS approach in predicting heat transfer was clearly shown in [9] for a test case of convection in an annular cavity with heated external cylindrical surface and adiabatic disks. In some papers (e.g. [10, 11]), numerical simulations were carried out using hybrid RANS/LES approach.
In this paper, we present the results of test simulations of the flow and heat transfer in a rapidly rotating cavity investigated in [12]. The study is aimed at testing the ability of a RANS/LES approach to predict accurately the flow structure in the cavity and heat transfer on the disk surface. To assess the computational model effect on the solution, various vortex-resolving techniques are considered along with evaluation of solution grid-sensitivity. The simulation results are compared with each other and with experimental data [12] and RANS computations [13].

2. Problem definition and numerics

2.1. Experimental setup and flow conditions

The present numerical study was based on experiments by Owen et al. [12]; the experimental setup is illustrated in figure 1a. Main proportions of the cavity were as follows: the outer radius \( b = 371 \) mm, the inner radius \( a = 150 \) mm, the cavity width \( s = 75.2 \) mm (\( a/b = 0.40, s/b = 0.20 \)). One of the cavity side walls (Disk 2 in figure 1a) was radiantly heated by a stationary electric heater; several RDF fluxmeters and thermocouples were mounted on the cavity-side surface of the disk. The opposite wall (Disk 1) was made transparent to allow LDA measurements of the flow velocity. The cooling air entered the cavity through an annular channel with radial gap \( \Delta r = 4.8 \) mm; radial vanes were attached to the upstream and downstream radial faces of the inner cylinder to ensure solid-body rotation of the incoming air. The cavity, as well as the air inlet and outlet channels rotating together with it, were enclosed in a stationary tight casing with some overpressure maintained inside.

![Figure 1. (a) Scheme of rotating cavity experimental setup [12] and (b) plot of the heated disk excessive temperature \( \Delta T = T - T_{in} \), as measured in [12] for Exp. No.6.](image)

It’s worth mentioning that, since the RDF fluxmeters measured the total heat flux including the convective and radiative ones, an approximate correction was introduced in [12] to obtain the convective heat flux data. However, that correction was not specified in details and only an estimation of the resulting Nusselt number uncertainty, sound as \( \Delta Nu = 50 \), was given in [12]. So, the experiments with relatively low typical values of Nusselt number (e.g., below 100) could not be considered as a fully reliable basis for comparison. As well, for most of experiments reported, the cavity rotation speed was not given in [12] that impeded reconstruction of the flow conditions. Under these circumstances, experiment No.6 (as defined in [12]) was selected for the present study, especially because that case was simulated numerically in [13].

The flow conditions in Exp. No.6 were defined as follows: the cavity rotation speed \( \Omega = 165 \) rad/s, Rossby number \( Ro = W/\Omega a = 0.173 \) (that yields the inlet flow velocity \( W = 4.3 \) ms), the inlet Reynolds number \( Re_{in} = 2W\Delta r/\nu = 2.95 \times 10^4 \), and the rotational Reynolds number \( Re_\Omega = \Omega b^2/\nu = 1.63 \times 10^6 \) (here \( \nu \) is the kinematic viscosity at the cavity inlet). Since neither reference pressure nor temperature were specified in [12], in the present study the inlet temperature, \( T_{in} \), was supposed to be 300 K, and the operating pressure was adjusted so that to fit the above non-dimensional criteria. The measured radial
distribution of the heated disk surface temperature and its approximation used in the computations are shown in figure 1b.

2.2. Computational grids
The flow domain and the computational grid structure are illustrated in figure 2. Main dimensions of the cavity correspond to the experimental rig [12] as described above; the cobs’ geometry was retrieved from the sketches given in [13]; the length of the inlet annular channel was 75 mm.

Three grids were used for the computations. The first one (hereafter referred to as grid G1) is shown in figure 2. It consisted of 2.16 million cells with 5400 quadrilaterals in the meridional plane and 400 divisions along circumference. In the middle of the cavity the cell size was about 3×3×4 mm. Near the side walls (discs) the grid spacing was 0.09 mm that resulted in the mean value of the normalized wall distance \( y^+ = 0.8 \) and the maximum value twice as much. Normally, such near-wall grid resolution is quite acceptable for Low-Re turbulence modelling. However, to achieve a better reproducing very thin Ekman layers (with the characteristic thickness, \( (\nu \Omega)^{1/2} \), about 0.3 mm), grid G1 was further refined close to the discs so that to obtain the near-wall grid spacing of 0.04 mm; this refined mesh (G2) had a total of 2.5 million cells. Finally, taking into account known irregularities of the flow under consideration, one more grid (G3) was obtained from grid G1 via halving the cell size in all three directions throughout the domain; the resulting grid size was 17.3 million cells.

2.3. Mathematical model and computational aspects
The computations were performed using Ansys Fluent 19 software. Among various turbulence modelling options available in Ansys Fluent, two distinct vortex-resolving approaches were used in the present study, namely unsteady RANS and LES. The RANS computations were carried out using grid G2 and the \( k-\omega \) SST turbulence model [14] with and without the rotation-curvature correction [15].

Most LES computations were performed using so-called zonal RANS-LES approach that implies manual division of the flow domain onto adjoining RANS and LES zones. In the present study, the RANS zone covered the annular channels and a small part of the cavity close to the inner cylinder (see figure 2c) whereas the other part of the cavity was treated as LES zone. The dedicated Vortex Method was applied at the RANS-LES interface to create reasonable pseudo-turbulent velocity fluctuations in the flow entering LES zone on the base of the approaching flow RANS solution. This strategy was intended to provide (a) an adequate RANS simulation of the flow close to the cavity inlet (where the grid was too coarse for LES prediction of turbulent spread of the incoming annular jet) and (b) switching to LES mode in the principal part of the cavity where this approach seemed to be more reliable.
In the framework of this zonal RANS-LES approach, four computations were carried out. Namely, a combination of the $k$-$\omega$ SST model [14] in RANS zone and the WALE subgrid turbulence model in LES zone was tested using the three grids created. Another variant comprising SST model in RANS zone but no subgrid model in LES zone was used with grid G3 (the finest one). Additionally, the “pure” LES (WALE) computation was performed using grid G3; in this case the inlet annular channel was removed and the vortex generator was applied directly at the cavity inlet. All the cases considered are summarized in table 1.

| Case | Model description | Grid |
|------|-------------------|------|
| 1    | Unsteady RANS $k$-$\omega$ SST | G2   |
| 2    | Same with rotation-curvature correction | G2   |
| 3    | Zonal RANS-LES (SST-WALE) | G1   |
| 4    | Same | G2   |
| 5    | Same | G3   |
| 6    | Zonal without subgrid model in LES zone | G3   |
| 7    | LES-WALE without RANS zone | G3   |

In the present study, the cooling air was treated as perfect gas, the viscosity and thermal conductivity were defined as piecewise-linear functions of temperature. Similar to [13], radial distribution of the heated disk temperature (defined via quadratic approximation of the measured values, see figure 1b) was applied as the boundary condition for that wall; other walls were treated as adiabatic. The problem was solved in the reference frame rotating together with the cavity; uniform axial flow was applied at the domain inlet section.

The pressure-based Fluent solver was used along with the SIMPLEC method of pressure-velocity coupling, the body-force weighted scheme of pressure interpolation, and the bounded second order implicit time-stepping. The second order upwind spatial discretization was used in RANS zone whereas in LES zone the scheme switched to bounded central differencing. The time step $\Delta t = 0.5$ ms was adopted for coarser grid computations but $\Delta t = 0.2$ ms was used with the finest grid G3. The resulting convective Courant number, CFL, was below unity (mostly below 0.5) in the principal part of the cavity. At that, close to the inner cylinder (especially near the cavity inlet and outlet sections) the local values of CFL exceeded 20.

Simulation of one second of physical time using 60 CPU cores took about one week for the finest grid (G3) cases and half a day for the coarser grids G1 and G2. Typically, it was necessary to simulate about two seconds of the process to arrive at statistically steady-state flow regime (starting from a previous solution) and up to four seconds more to collect representative unsteady flow statistics. To reduce the statistical noise, the time-averaged parameters of interest were additionally averaged over several meridional sections.

3. Results and discussion

As an example of the simulation results, figure 3 presents a snap shot of relative velocity vectors (colored by the velocity magnitude in m/s) and excessive temperature, $\Delta T = T - T_m$, obtained in the framework of zonal RANS-LES approach using grid G2 (Case 4 in table 1). Similar flow fields obtained using the finest grid G3 (Case 5) are shown in figure 4.

Though the mid-plane velocity field shown in figure 3a is rather complicated, one is able to extract several cyclonic (co-rotating with the cavity) and anti-cyclonic (counter-rotating) vortices that are highlighted by superimposed “streamlines”. In all, two pairs of global vortices with cyclonic and anti-cyclonic circulation are clearly seen in figure 3a, that is consistent with observation [12] based on spectral analysis of LDA velocity measurements.
In the mid-plane temperature map (figure 3b) one can observe radial “arms” of relatively cold fluid that correspond approximately to the radial outflow regions between the cyclonic and anti-cyclonic global vortices (see figure 3a). However these “arms” are highly disturbed by the velocity fluctuations and they actually disappear until reaching the shroud. Figure 3c evidences that in the principal part of the cavity, the radial variation of the temperature is rather weak as compared to the axial one.

Comparing figures 3 and 4 one can conclude that usage of a much finer grid G3 didn’t affect the general flow structure. In particular, two pairs of global vortices still persist in figure 4a, and typical temperature and velocity levels are the same in both figures. On the other hand, as expected, the finer grid ensured resolution of smaller-scale vortices that resulted in more pronounced chaotic component of the flow. In particular, the cold radial “arms” in figure 4b seem to be nearly broken down by resolved velocity fluctuations.

Figure 3. Instant velocity vectors (a) and contours of excessive temperature $\Delta T$ (K) (b,c) over the cavity mid plane (a,b) and meridional section (c) obtained on the base of zonal RANS-LES approach using grid G2 (Case 4 in table 1).

Figure 4. Same obtained using grid G3 (Case 5 in table 1).

Figure 5 shows distributions of the non-dimensional heat transfer coefficient, the Nusselt number, obtained with two grids. Following [12], the local Nusselt number is defined as $Nu = q_w r (k \Delta T_w)^{-1}$, where $q_w$ is the wall heat flux, $\Delta T_w$ is the disk excessive temperature at current radius $r$ (see figure 1b), $k$ is the fluid thermal conductivity at the inlet temperature $T_{in}$. Quite similar to the grid refinement effect on the flow structure, the Nusselt number obtained using finer grid G3 (figure 5b) is more chaotic, with sharp maxima and minima, but the large-scale features and typical levels are roughly the same.
Figure 5. Instantaneous distributions of Nusselt number over the heated disk obtained from Case 4 (a) and Case 5 (b) computations.

In figure 6, radial distributions of Nusselt number obtained using the two grids are compared with the measurement results [12]; in addition to the mean data, instantaneous profiles extracted from four meridional sections are presented in the plots. Consistent with the previous observations, in the finer grid solution (figure 6b) the instant Nusselt number profiles are highly irregular and the fluctuations’ magnitude even exceeds the mean values. Unlike that, in figure 6a (coarser grid) the four instant profiles of $\Nu$ are more or less the same and close to the mean one except rare solitary perturbations. In particular, a noticeable peak at $r/b=0.9$ in figure 6a is a trace of small red spot on the left side of figure 5a that was occasionally crossed by the meridian selected. It is worth mentioning also that in the innermost region of the disk, at $r/b<0.6$, the instant values of $\Nu$ happen to be negative, i.e. the disk is sometimes heated from a cloud of hotter air that follows toward the center along the Ekman layer. As to the mean profiles of $\Nu$, the two cases yielded the results very close to each other but with serious disagreement with experimental data [12].

Figure 6. Instant and averaged radial distributions of Nusselt number obtained from Case 4 (a) and Case 5 (b) computations along with experimental data [12].

Figure 7 presents the mean radial profiles of the Nusselt number and the mid-plane rotational velocity (in absolute frame of reference): the data obtained using all the models tested (see table 1) are compared with the experimental data [12] and with the results of unsteady RANS ($\text{RNG } k-\varepsilon$) computations [13].
Figure 7a evidences that the heat transfer rate was underestimated considerably (roughly 2 to 3 times) in all computations. Apart from this disappointing quantitative disagreement, the best results (higher values of $Nu$) were obtained using unsteady RANS approach without any noticeable difference between the $k$-$\omega$ SST (Case 1) and RNG $k$-$\varepsilon$ predictions. It’s well known however that in the “standard” formulation, these models do not provide an adequate simulation of turbulent mixing in rapidly rotating flows. Surprisingly, activation of the dedicated rotation-curvature correction [15] (Case 2) resulted in the worst prediction of the heat transfer rate. So we can conclude that the observed “success” of the RANS models was rather deceptive and, presumably, resulted from some error compensation. RANS-LES computations with coarser grids G1 and G2 (Case 3 and 4) yielded nearly same results as SST-CC model (Case 2). Usage of the finer grid G3 resulted in somewhat higher values of $Nu$ with insignificant difference between the three models tested (Cases 5 to 7).

![Figure 7](image-url)

**Figure 7.** Radial distributions of Nusselt number (a) and mid-plane rotational velocity (b) obtained here (Cases 1 to 7 in table 1) along with measurements [12] and RNG $k$-$\varepsilon$ computations [13].

A comparison of the mid-plane rotational velocity profiles shown in figure 7b leads to similar general conclusions. All the computations yielded more or less close profiles though the data obtained using the RANS models without rotation-curvature correction were slightly apart. However in all, the observed difference between numerical results is much less than the disagreement with experimental data [12]. The systematic underestimation of the rotational velocity lag (i.e. the speed of core precession with respect to the cavity) seems to be consistent with underestimation of the wall heat transfer. However the reasons for this underestimation still remain undisclosed.

4. Conclusions
The flow and heat transfer in a rotating annular cavity with one heated disk and axial throughflow has been simulated using various vortex-resolving models. Two pairs of cyclonic and anti-cyclonic global circulations have been detected in the flow predicted that is consistent with available experimental data for the test case considered. Despite LES computations using a considerably refined mesh have yield much more intensive fluctuations of the wall local heat flux as compared to coarser grid predictions, the mean radial distributions of the disc-surface Nusselt number obtained in different computational cases are more or less close to each other but all of them underestimate the disk-surface heat transfer significantly. As well, the computed mid-plane mean velocity profiles exhibit a considerable underestimation of the rotational velocity lag. Further investigations are needed to reveal the reasons of “stable” disagreement between the simulation and measurement results for disc-surface heat transfer.
Acknowledgments
The study was supported by the Russian Foundation for Basic Research under grant no. 20-08-01090.

References
[1] Harmand S, Pelle J, Poncet S and Shevchuk I V 2013 Int. J. Thermal Sciences 67 1–30
[2] Bohn D, Deuker E, Emunds R and Gorzelitz V 1995 J. Turbomachinery 117 175–83
[3] Bohn D, Deutsch G, Simon B and Burkhardt C 2000 Proc. ASME Turbo Expo (Munich) GT2000-280
[4] Ivanov N G, Ris V V, Smirnov E M and Smirnovskii A A 2005 Proc. 3rd Int. Conf. Advanced Computational Methods in Engineering (Ghent) p 9
[5] Bohn D, Bouzidi F, Kitanina E, Ris V V, Smirnov E M, Burkhardt C and Wolff M W 2002 9th Int. Symp. Transport Phenomena and Dynamics of Rotating Machinery (Honolulu) p 8
[6] King M P and Wilson M 2005 Numer. Heat Transf. A 48 529–45
[7] Tian S, Tao Z, Ding S and Xu G 2008 Int. J. Heat Mass Transfer 51(3-4) 960–8
[8] Wroblewski W and Fraczek D 2014 Arch. Mech. 66(5) 343–64
[9] Sun X, Linblad K, Chew J W and Young C 2007 J. Eng. Gas Turbines Power 129(2) 318–25
[10] Smirnov P E, Kapetanovic S, Braaten M E, Egorov Y and Menter F R 2009 ASME Paper GT2009-59621
[11] Atkins N R and Kanjirakkad V 2014 Proc. ASME Turbo Expo (Dusseldorf) GT2014-27174
[12] Owen J M and Powell J 2006 J. Eng. Gas Turbines Power 128(1) 128–34
[13] Owen J M, Abrahamsson H and Lindblad K 2007 J. Eng. Gas Turbines Power 129(4) 893–900
[14] Menter F R, Kuntz M and Langtry R 2003 Turbulence, Heat and Mass Transfer 4 (Begell House) pp 625–32
[15] Smirnov P and Menter F 2009 J. Turbomachinery 131(4) 041010