Large Eddy Simulation of Spanwise Rotating Turbulent Channel and Duct Flows by a Finite Volume Code at Low Reynolds Numbers

Kursad Melih Guleren* and Ali Turan

University of Manchester, School of Mechanical, Aerospace and Civil Engineering, M60 1QD Manchester, UK
M.Guleren@postgrad.manchester.ac.uk
A.Turan@manchester.ac.uk

Abstract. The objective of this study is to show the highly complex features of rotational turbulent flow using a widely known finite volume code. The flow subjected to an orthogonal rotation is investigated both qualitatively and quantitatively in a three-dimensional channel and a duct using FLUENT. The predictions of rotational flow calculations, presented for low Reynolds numbers, both in channel and duct are in good agreement with the DNS predictions. It is of interest to present the capability of the code for capturing the multi-physics of internal flow phenomena and to discuss the Coriolis effects for two rotational rates. The results show that FLUENT is able to predict accurately first and second order turbulent statistics and it also captures the proper secondary flow physics which occur due to rotation and the geometry itself. These results are very encouraging for the simulation of the flow in a centrifugal compressor, which is the main goal of the authors in the long term.

1 Introduction

It is well known that investigation of the turbulent fluid motion is a challenging research area; neither an analytical solution exists nor it can be exactly defined mathematically. Its complexity is generally explained with its unsteadiness, three-dimensionality, dissipative and diffusive features. In addition, it contains a broad spectrum, which is formed by various size of eddies. For example, scales of these eddies can be of the order of the size of the flow geometry and of the size of Kolmogorov scale, which is known as the smallest scale. Even without rotation, turbulent is certainly a multiscale process. Combining the effects of rotation with turbulence makes the flow physics more interesting, however more complex, and difficult to analyze either experimentally or numerically. It was confirmed by previous studies that rotation changes not only the mean flow but also the turbulence field itself. Although, there exist a wide range of studies in literature as to how and why the multiphysics of these flows are affected depending on the Reynolds and rotation numbers,

* Permanent address: Cumhuriyet University, Dept. of Mech. Eng., 58140, Sivas, Turkey.

V.S. Sunderam et al. (Eds.): ICCS 2005, LNCS 3516, pp. 130 – 133, 2005.

criteria still remains to be formulated clearly for practical industrial flow applications including the centrifugal compressor.

The present study analyzes a turbulent rotating channel flow at a low Reynolds number of Re=2800 for two rotation numbers of Ro=0.1 and Ro=0.5 ($Re=U_b h/\nu$, $Ro=2\Omega h/U_b$, where U_b is the bulk velocity, h is the half width of the channel, ν is the kinematic viscosity and Ω is the rotation rate of the flow). In addition, the turbulent duct flow is investigated at a low Reynolds number of Re=4410 for two rotation numbers of Ro=0.013 and Ro=0.053 ($Re=U_b D/\nu$, $Ro=2\Omega D/U_b$, where D stands for the hydraulic diameter of the duct).

2 The Model

For the channel flow, dimensions of the geometry are L_x=6.4h, L_y=2h, L_z=3.2h. For the duct flow, spanwise and radial lengths are set to be equal to $L_y=L_z=D$ while the streamwise length is taken as L_x=6.28D. The flow is assumed to be fully developed, isothermal, incompressible and rotating at a fixed positive angular velocity parallel to the spanwise direction, $\mathbf{\Omega}=(0,0,\Omega)$.

The numerical calculations were performed using the development version of the general-purpose code FLUENT V6.2 [1] using the Dynamic Smagorinsly-Lilly Model [2],[3]. The code is based on a finite-volume method with an unstructured grid algorithm. The LES incorporates 2nd order central differencing for the diffusive and convective terms for the channel flow calculations and 3rd order MUSCL for duct flow calculations. A fully second-order implicit scheme is applied for temporal discretization while the PISO algorithm and PRESTO! scheme are employed for the velocity-pressure coupling and pressure interpolation, respectively.

The computational domain is formed by $66 \times 66 \times 66$ and $66 \times 60 \times 60$ cells (in the x,y and z-directions) for channel flow and duct flow, respectively. The computational grid is equally spaced along the homogenous directions (x and z-directions for channel and x-direction for the duct) and stretched non-uniformly between the solid walls [from y=0 (bottom wall) to y=2h (top wall) in the channel, from y=0 (bottom wall) to y=D (top wall) and from z=0 (lateral wall) to z=D (lateral wall) in the duct]. Non-slip boundary conditions and periodic boundary conditions were applied for the walls and homogenous directions, respectively. Constant mass flow rate was assumed in the flow directions rather than constant pressure drop.

3 Results and Discussion

Fig. 1. shows the distribution of mean and turbulent intensities for low and high rotational rates. For both cases, excellent agreement for the mean velocity was found with the DNS data [4] except for a slight increase near the pressure side (y=0) at the low rotation rate. For this case, radial and spanwise turbulent intensities are remarkably under-predicted near the pressure side, but they gradually approach the DNS data through center of the channel. Spanwise intensity, which is mainly responsible for turbulent kinetic energy, is under- and over-predicted near the pressure and suctions sides, respectively. For the high rotational case, similar trends are obtained however the discrepancies and the range that is affected are smaller in this case.

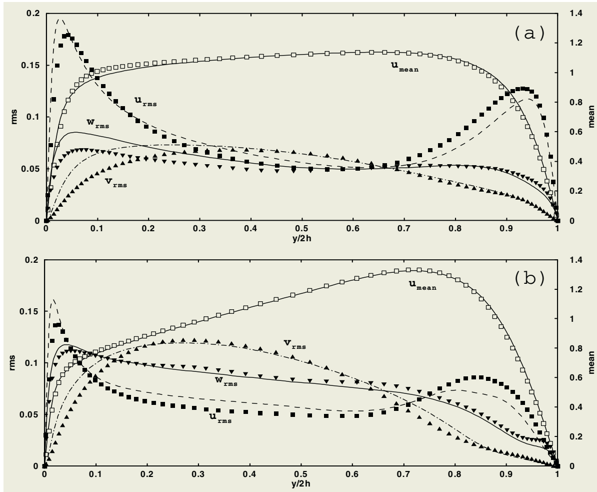

Fig. 1. Mean velocity and turbulent intensity profiles for the rotating channel flow at Ro=0.1 (a) and at Ro=0.5 (b). Present LES results, shown with symbols, are compared with DNS results of Kristoffersen and Andersson [2], shown with lines. Values are normalized by bulk velocities

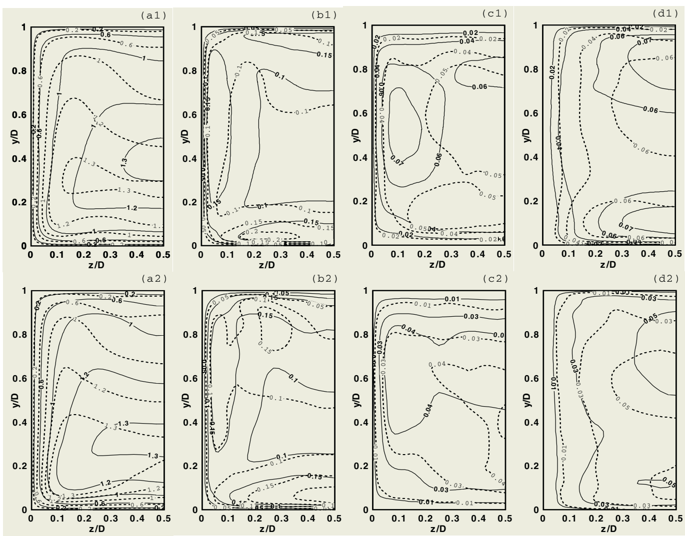

Fig. 2. Mean velocity (a1,a2) and turbulent contours (b1,b2 for streamwise, c1,c2 for normalwise, d1,d2 for spanwise directions) for rotating duct flow at Ro=0.013 shown with lines and at Ro=0.053 shown width dashes lines. While the top figures represent the DNS results of Gavralakis[16], the bottom figures represent the present LES results. Due to symmetry, half of the duct is shown

Spatial distribution of the mean velocity and turbulent intensities for half of the square duct is also shown for two rotational cases in Fig. 2. The peak region of the predicted mean velocity (a2) has a tendency to shift towards the bottom corner; otherwise, the LES predictions are in good agreement with the DNS data [5] for both rotational cases. Streamwise turbulent intensity (b2) seems to be consistent with the DNS, but the remaining intensities (c2,d2) are remarkably under-predicted. However, the under-prediction for Ro=0.053 is less than that for Ro=0.013. Notwithstanding these discrepancies, our results are similar to those of Palleres and Davidson [6]. Considering the increase in rotational number, mean velocity and turbulent intensity decrease near the suction side and increases towards the pressure and lateral side of the duct. The normal and spanwise turbulent intensities are observed to be reduced near the suction side and become enhanced near the pressure side.

4 Conclusion

Spanwise rotating channel and duct flow were investigated at low Reynolds numbers using LES. Although there are some discrepancies at low rotational numbers, the results are generally in good agreement with DNS predictions. These discrepancies are thought to be caused primarily by the SGS model incorporated in FLUENT. Additionally, the highest accuracy for the numerical schemes in the code is third order: it is well known that such attributes might provide another source for discrepancies.

Concerning future studies, plans are already underway to test different SGS models, including dynamic kinetic energy SGS model [7] and the wall-adapting local eddy-viscosity (WALE) model [8], in order to understand the performances of these SGS models for rotating channel and duct flow problems.

References

1. FLUENT 6.1 USER GUIDE Fluent Inc. Lebanon, USA (2001)
2. Germano, M., Piomelli, U., Moin, P. & Cabot, W. H. A dynamic subgrid-scale eddy viscosity. Phys. Fluids 7 (1991) 1760
3. Lilly, D. K. A proposed modification of the Germano subgrid-scale closure method. Phys. Fluids 4 (1992) 633
4. Kristoffersen, R. & Andersson, H. I. Direct simulation of low-Reynolds-number turbulent flow in a rotating channel. J. Fluid Mech. 256 (1993) 163
5. Gavralakis, S. Direct numerical simulation (DNS) of the Rotating Square Duct flow at a low turbulent Reynolds number. http://lin.epfl.ch/index2.php/link/staff/id/43 (results not published yet)
6. Palleres, J. & Davidson, L. Large-eddy simulations of turbulent flow in a rotating square duct. Phys. Fluids 12 (2000) 2878
7. Kim, W. and Menon, S. Application of the localized dynamic subgrid-scale model to turbulent wall-bounded flows. J.Fluid Mech. 35th Aerospace Sciences Meeting & Exhibit, Reno, NV, (1997) AIAA Paper 97-0210
8. Nicoud, F. & Ducros, F. Subgrid-scale stress modeling based on the square of velocity gradient tensor. Flow, Turb. Comb. 62 (1999) 183