RANS COMPUTATION OF RIBBED DUCT FLOW USING FLUENT AND COMPARING TO LES

Máté M., Lohász∗&/ Ákos Csécs∗
+ Department of Fluid Mechanics, Budapest University of Technology and Economics, Budapest
* Von Kármán Institute for Fluid Dynamics

& LES part of this article is done in the frame of the PhD research of the first author under supervision of Patrick Rambaud, Carlo Benocci

Abstract
The commercial code Fluent 6.1 was used in past for computation of a square section duct flow with high blockage perpendicularly arranged square section ribs, using LES. This gives a database for validation of RANS turbulence models. The RNG k-epsilon model was assessed against the LES result, using profiles of stream-wise and vertical velocities. The secondary flow characteristics of the two methods was compared using streamlines of the in plane velocities. The RNG model does not satisfactory predict all of the flow features.

Introduction
Flow inside a ribbed duct has a high industrial interest. The internal cooling channels of turbine blades usually have such geometry. Because of this high industrial interest big amount of measurements were carried out investigating integral quantities (pressure drop and averaged Nusselt number) and fewer on detailed flow and heat transfer characteristics. Other field of research is the computational investigation. The flows which have industrial interest are turbulent. Turbulence is nowadays still difficult to resolve by the computational grid (DNS) Jiménez 2002. The turbulent motion has to be modelled partially or completely.

The most common way of partially modelling is the Large-Eddy Simulation (LES), Piomelli 1999, where the large, energy containing motions are resolved by an unsteady computation, and the smaller ones are modelled using usually a simple algebraic relationship. For the fully three dimensional flow field in the ribbed duct the computational requirements are still high. Murata et al. 2000, Watanabe et al. 2002, and Lohász et al. 2003b published computations using LES.

The computation using complete modelling has much less computation requirement and already in the last century big amount of work was published, for a review see Iacovides 1999. In this frame the Reynolds averaged Navier-Stokes equation is solved looking for a steady solution, using different kind of semi-empirical equations for the modelling of the terms appearing thanks to the nonlinearity of Navier-Stokes equation. Because of the continuous progress in the modelling of the Reynolds Averaged Navier-Stokes equations (RANS) these models are used to compute the very challenging ribbed duct flow with its difficult turbulence physics. Recently Manceau et al. 2000, Ooi et al. 2002, Ooi 2002b, Iaccarino et al.2002 and Bredberg et al. 2002 published their results using
Nusselt number and mean velocity profiles for validation. They could compute a whole ribbed duct with bend, which is nowadays unfeasible using LES. Ooi et al. 2002a presented impressive 3 dimensional flow features but without any validation. In the present work we present computation results using commercial code with built in Renormalization Group (RNG) k-epsilon model for a ribbed duct with high blockage, which was computed Lohász 2003a and Lohász et al. 2003b using LES and was found to contain strong three dimensional features. This was previously validated against the extensively experimental results of Casarsa et al. 2002.

The investigated flow is the fully developed flow in a square section duct with square section ribs successively mounted on the bottom wall perpendicular to the mean flow direction. The flow is considered incompressible with constant density and viscosity. The Reynolds number is 40000 with the hydraulic diameter of the duct and the bulk velocity. The rib to hydraulic diameter ratio (h/D) is 0.3 and the pitch to rib ratio (p/h) is 10.

**Computational method**

The segregated solver of the commercial code Fluent 6.1 was used to solve the Reynolds equations in one pitch length. In the transport equations second order upwind method with cell based gradient formulation and traditional slope limiter was used for the interpolation of convective fluxes, the pressure in the momentum equations was interpolated using the standard method. In span-wise direction (denoted by Z) the symmetry of the geometry has been used to reduce the computational requirements. In stream-wise direction (X) periodic boundary condition was used for velocity and pressure and a forcing term was added to the momentum equation to ensure the required mass flow rate.

For modelling the Reynolds stresses the RNG k-epsilon model as implemented in Fluent 6.1 was used with differential viscosity model for swirl dominated flows including pressure gradient effects with the proposed constants implemented as default in the program. For the wall modelling the enhanced wall treatment was used to enable low Reynolds number wall modelling.

The mesh was unstructured in the plane perpendicular to the Z axis, and contained 41563 quad cells (see Fig.1-2.). At the walls in normal direction refinement was used to enable the required unity cell size in wall units using 0.00025D as smallest wall normal cell size. In the core of the domain a cell size of 0.05D was reached using a maximum growth rate of 1.2. In span-wise direction a structured mesh of 35 cells was built, the proximity of the wall was refined. This grid satisfies the requirement of having at least 10 cells in the region of the boundary layer with lower turbulent Reynolds number than 200.

The computation can be treated as grid independent in the sense that the pressure drop determined on a coarser mesh (427584 cells) differed only by 0.4%.

The computation with 1454705 cells using single precision numbers was running in parallel for 3 weeks on a double processor 2GHz PC with 1.2 GB memory requirement to reach convergence of the number representation.
Results
The most important result from engineering point of view is the pressure drop. The non-dimensional value computed by LES is 0.257 and the RNG model gives the value of 0.137. The RNG turbulence model under-predicts the value by a factor of two. To understand this discrepancy the mean velocity components in the middle plane were compared to the LES result of Lohász 2003a. In Fig. 3. a big difference can be seen in the stream-wise velocity component. The high value region is close to the upper wall instead being above the rib. The flow direction is opposite in the middle of the rib, this is the consequence of the fact that the flow computed using RNG reattach on the top of the rib in a position upstream from the position where the LES predicts the
reattachment. Most probably the reason for this is the poor prediction of turbulence phenomena around the leading edge of the rib. The recirculation region between the ribs has different shape, the reattachment position is more downstream computed by RNG, and this is in correspondence with the lower pressure drop. The acceleration downstream of the reattachment point is predicted poorly again with the RNG model. This phenomena is similar to the well know problems of stream-wise velocity recovery studied on the case of backward facing step (see for example Bauer 2000).

Figure 3. Stream-wise velocity X=const. profiles, LES circle, RNG cross.

Considering the velocities perpendicular to the ribbed wall in the centre plane of the duct (Fig. 4.) it seems to be clear that the velocity predicted by RNG is lower, which means a lower three dimensional character of the flow.

Figure 4. Wall normal-wise velocity X=const. profiles, LES circle, RNG cross.

For the investigation of the three dimensional character of the flow streamlines of in-plane velocities were drawn at two different stream-wise position. In the Fig. 5-6. the most obvious difference between RNG and LES is that the circular counter-clock motion exist only in the LES result. It is not clear if this is a secondary flow induced by pressure difference or secondary flow induced by Reynolds stress anisotropy (like in duct flow). The second one is impossible to predict with linear eddy viscosity models like the used one. The difficulty of the prediction of the first kind could be explained by
the discrepancy in the prediction of the recirculation region between the ribs, resulting also a different pressure distribution.

![Figure 5](image)

**Figure 5.** Secondary flow and stream-wise velocity contours in X/h=0 a), b) and X/h=5 c), d) plane.

a) c) RNG, b), d) LES

The same difference in the stream-wise velocity can be seen as in the Fig. 3, the high value region with RNG is closer to the upper part of the duct, this is in correspondence with the fact that the vertical (Y direction) velocity in the direction of the bottom is lower (see Fig. 4.).

**Conclusion**

The strongly wall bounded three dimensional flow in high blockage ribbed duct was computed using the commercial code Fluent 6.1 with built in RNG k-epsilon model. The result was validated against a LES computation with the same flow configuration. The main feature of the stream-wise velocity was captured including the recirculation region between the successive ribs, but big discrepancies occurred in the values. The secondary flow characteristics were also compared, pointing out strong differences, the RANS approach under-predicts the secondary velocities.

For deeper understanding the comparison of Reynolds stresses and pressure terms would be needed, this is proposed for the future.

The computation effort of the RANS computation was quite high comparable to LES, because of the high number of cells and slow convergence. This could be strongly reduced with coarser mesh and lower level of convergence, but experience is needed to judge the result of such a simplification.

**References**


Watanabe, K. and Takahashi T. 2002. LES simulation and experimental measurement of fully developed ribbed channel flow and heat transfer. *Proceedings of ASME TURBO EXPO 2002 June 3-6, 2002 Amsterdam, The Netherlands*


Bredberg, J. and Davidson, L. 2002. Prediction of turbulent heat transfer in stationary and rotating U-duct with rib roughened walls. *In Engineering Turbulence Modelling and Experiments., 5*