Center for Turbulence Research
Annual Research Briefs 2003

Numerical and experimental investigation of the turbulent flow in a ribbed serpentine passage

By **Gianluca Iaccarino, Georgi Kalitzin† and Christopher J. Elkins‡**

1. Introduction and motivations

Modern gas turbine engines operate at high combustor outlet temperatures to achieve higher thermal efficiency and thrust. Turbine blades are exposed to these high-temperature gases and undergo severe thermal stress and fatigue. The design of highly efficient cooling systems for turbine blades has an enormous potential impact on engine development. Cooling devices are based on a secondary flow system built into each blade, as illustrated in Fig. 1. The secondary flow passages are extremely complicated consisting of one or multiple legs with turbulators (rib-roughened serpentines), holes connecting the secondary path to the external surface of the blade (film cooling), tube bundles, slots, etc. The geometrical complexity of these passages, however, is extremely challenging for the use of advanced simulation tools based on state-of-the-art three-dimensional CFD solvers. The generation of a computational grid (even using unstructured mesh technology) requires a considerable amount of time. As a result, the analysis of the cooling performance of the system is largely based on isolated sub-component simulations, simplified one-dimensional models and experimental correlations.

A large amount of research has been devoted to the analysis of the flow characteristics (in particular turbulence statistics) and heat transfer in channels with ribs. The effects of rib shape, pitch, height and inclination with respect to the incoming flow have been quantified from an experimental point of view. Numerical predictions of these flows are complicated because of the increased turbulence intensity near the ribs and its effect on the flow. In Reynolds-Averaged Navier-Stokes (RANS) approaches the turbulence intensity is strongly affected by the choice of the turbulence closure. RANS eddy viscosity models based on the linear Boussinesq relationship between Reynolds stresses and velocity gradients are the *de facto* standard in industrial numerical simulations. Previous investigations (Ooi *et. al.* 2002a; Iacovides & Raisee 1999) have illustrated the predictive capabilities of such models in turbomachinery cooling systems components: smooth channels, channels with ribs, 180 degrees turns (U-bends), etc. In particular, it has been shown that the flow is strongly three-dimensional and the presence of oblique ribs generates strong coherent vortices in the domain (Ooi *et. al.* 2002b). It must be noted that most of these simulations are performed under the assumption that the flow is periodic in the streamwise direction and, therefore, only a small section of the passage is effectively simulated.

Several experimental datasets are available to evaluate the accuracy of these simulations (Han *et. al.* 1985; Rau *et. al.* 1998); typically the measurements are taken in devices that include many ribs in between duct U-bends and the data are collected at different locations to verify that periodicity conditions are indeed applicable. Although these comparisons between experiments and computations are invaluable in improving

---

† CITS
‡ Mechanical Engineering Dept., Stanford University

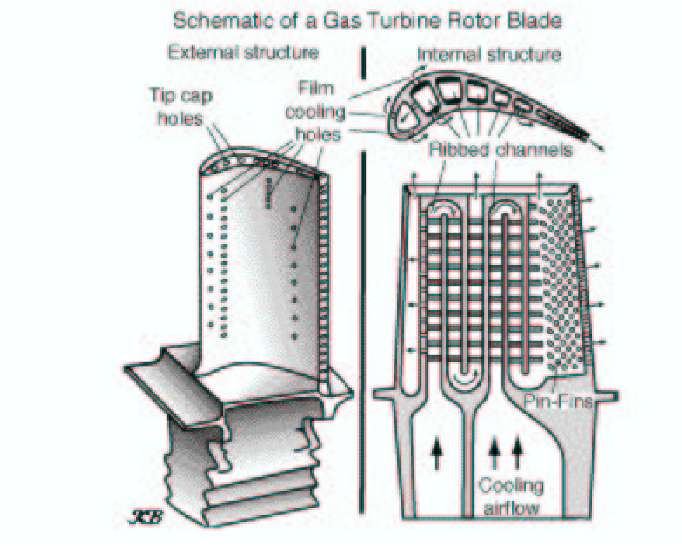

FIGURE 1. Sketch of the Cooling System of a Turbine Blade (Cumpsty 1997).

the understanding of the physics and in providing accurate model validation, they do not represent the wealth of complexity and interactions present in more realistic configurations, like a multiple-leg serpentine.

Unfortunately the computational cost of such simulations becomes substantial. For a single three-dimensional rib-roughened passage it might be necessary to generate a grid with half million cells (Ooi *et. al.* 2002b) to completely resolve the flow structures. In a serpentine with thirty ribs and two 180 degree turns, the mesh size should be in the order of 30 million cells. In addition, it is necessary to obtain extensive experimental datasets to quantify clearly the accuracy of the simulations.

It is clear that a novel numerical and experimental technique for investigating flows in complex geometries would be extremely beneficial to the development and improvement of the predictive tools used in the design.

In this paper, the turbulent flow in a serpentine with oblique ribs is investigated experimentally and by numerical simulations. The measurements are carried out by using Magnetic Resonance Velocimetry (MRV) (Elkins *et. al.* 2003) and the simulations using the Immersed Boundary (IB) technique (Iaccarino & Verzicco 2003; Kalitzin & Iaccarino 2002). A brief description of these two approaches is reported in following sections. The results are reported in terms of velocity distributions in various planes in the serpentine; differences between measurements and simulations are presented qualitatively and quantitatively. The study of the discrepancy allows us to identify areas of needed improvements in the turbulence modeling.

2. Ribbed serpentine

The serpentine used in this study is represented in Fig. 2; the model was drawn in AutoCAD and fabricated using a 3D System 250-50 SLA machine. The serpentine has a square cross section of height $H = 20mm$ and ten staggered oblique ribs on the top and bottom walls. The rib height is $0.1H$ and the pitch (distance between two successive ribs) is $0.6H$; the rib angle is 45 degrees. A fully developed pipe flow enters the first leg of the

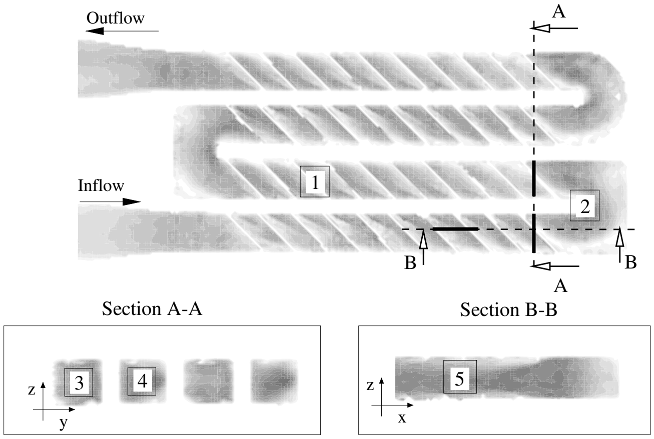

FIGURE 2. Sketch of the cooling passage investigated and the region where the experiments and computations have been compared.

serpentine through a converging section yielding a uniform velocity profile. Two Reynolds numbers have been investigated, 8,000 and 50,000 based on the serpentine height and the bulk velocity; in the present paper only the results at the low Reynolds number are reported.

3. Experimental technique

Magnetic Resonance Velocimetry, MRV, has been used to measure the mean velocity components in the entire serpentine. A massive amount of data, corresponding to about 2 million locations with $1mm$ resolution ($0.05H$) has been collected. The measurement technique is described in detail in Elkins *et. al.* (2003) where a simple pipe flow and a serpentine similar to the one studied here (Fig. 2) are investigated.

The MRV technique is a non-invasive experimental method for measuring mean velocities and is implemented in medical magnetic resonance imaging systems. MRV has been used for many low-Reynolds number investigations (Fukushima 1999), such as physiologic, multiphase, and porous media flows. It has also been used in a few simple turbulent flows such as pipe and jet flow. Our current focus is in high Reynolds number turbulent flows in complex internal geometries with the objective of measuring the time-averaged three-component velocity field in order to provide physical insights and an extensive database for CFD validation. It should be mentioned that the MRV technique is compatible with the rapid prototyping processes of stereolithography and fused deposition machining. Utilizing current RP technology highly complex flow models can be designed and manufactured in only a few days, and the MRV measurements, typically producing over 1 million velocity vectors, can be completed in 30 minutes.

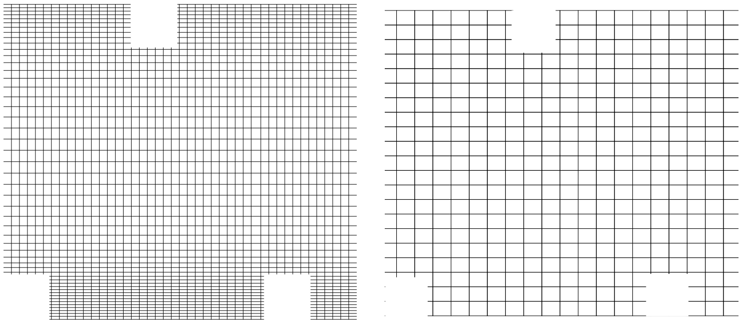

Figure 3. Resolution used in the coarse grid simulations (left) and experiments (right) in the region 5 (see Fig. 2).

4. Numerical technique

The numerical simulations presented in this work are based on the Immersed Boundary (IB) technique. The method is presented in detail in Iaccarino & Verzicco (2003) and Kalitzin & Iaccarino (2002); only a brief description is reported herein.

In the IB method Cartesian non-uniform meshes are used and forcing terms are added to the governing equations to enforce boundary conditions (no slip velocity, for example) on surfaces not aligned with the grid lines. The computational code used is based on the steady-state incompressible RANS equations closed with either a one-equation (Spalart & Allmaras 1994) or a two-equation model (Kalitzin & Iaccarino 2002). A second-order implicit discretization is used within a SIMPLE pressure-velocity coupling algorithm; the turbulence equations are solved in a segregated manner. Multigrid is used to accelerate the convergence. The IB forcing is based on a tri-linear reconstruction (Iaccarino & Verzicco 2003) for the velocity components and the turbulent scalars.

The geometry definition is based on the STL file format and, therefore, the same AutoCAD model used to build the experimental configuration has been used to obtain a simplified CFD computational model. The computational grid and the IB interpolation stencil is generated automatically and the pre-processing time to start the computation is negligible.

5. Results and discussion

The geometrical configuration is shown in Fig. 2. The CFD domain includes *only* the first three legs of the serpentine. The simulations have been performed using computational grids ranging from half million to forty millions elements. An example of the coarse grid resolution of the CFD mesh is shown in Fig. 3. The underlying grid for the MRV experiments is shown in the same figure.

The fine grid simulations are compared to the experimental data in the areas indicated in the sketch of Fig. 2. In Fig. 4 and 5 two horizontal planes are reported midway through the ribs and halfway through the passage, respectively. In Fig. 7 and 8 two vertical sections are reported with the streamwise direction pointing away from the page and toward it, respectively. Finally, in Fig. 6 a vertical longitudinal plane showing the streamwise flow (from left to right) is reported.

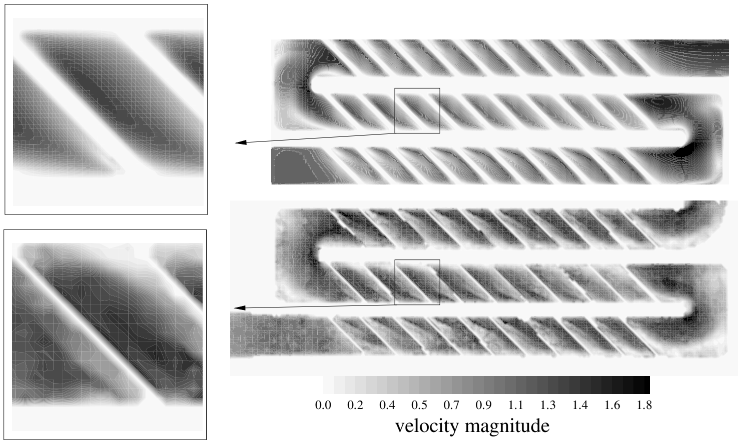

FIGURE 4. Comparison between simulations (top) and experiments (bottom) in region 1 (see Fig. 2): horizontal plane corresponding to a cut through the ribs.

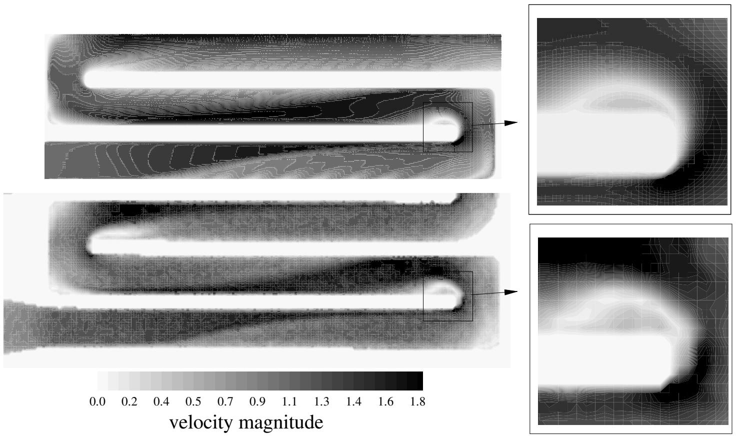

FIGURE 5. Comparison between simulations (top) and experiments (bottom) in region 2 (see Fig. 2):. horizontal plane corresponding to a cut at the mid height of the passage.

The general qualitative agreement between experiments and simulations is remarkable. In particular, the secondary flows in between the ribs described in Ooi *et. al.* (2002b) are well represented. The flow in between the U-bends shows a repeating patterns after about three or four ribs demonstrating the importance of the complete simulation as opposed to a periodic-rib channel study.

The simulations capture the most dominant effects illustrated in the experiments:

(*a*) A strong three-dimensionality, as indicated by the mid-height plane. High velocity regions oblique to the direction of the flow (and orthogonal to the rib orientations) indicate the presence of strong streamwise vortices;

(*b*) Recirculation bubbles in the downstream region of the 180 degree bends; in particular these features are in remarkable good agreement (in terms of length and maximum reverse velocity) with the measurements;

(*c*) A massive separation behind each rib, Fig. 7. Again the length and the velocity

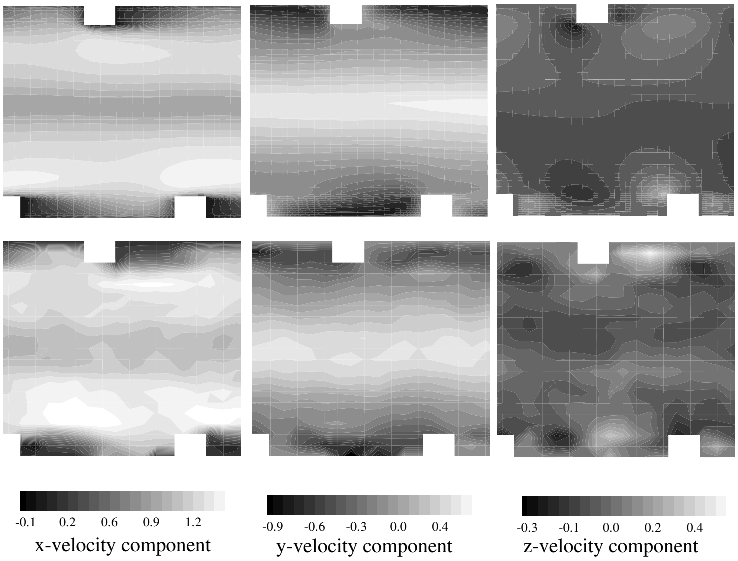

FIGURE 6. Comparison between simulations (top) and experiments (bottom) in region 5 (see Fig. 2): vertical plane corresponding to a longitudinal section before the U-bend.

magnitude are in good agreement, even if the resolution in the experiments is limited (cfn. Fig. 3).

(*d*) High speed regions atop of each rib with high shear and turbulence (not reported).

In addition, they clearly indicate that the flow in the straight sections is accurately represented and the largest discrepancies are around the U-bend regions. This is a well known limitation of the (linear) eddy viscosity models used in the RANS simulations and it does not appear to be linked to the Cartesian nature of the flow solver.

The wealth of information provided by the MRV technique allows a clear identification of the accuracy of the simulation. In particular, it can be used to demonstrate the limitation imposed by the turbulence model. The three components of velocity are reported for various flow regions in Fig. 6, 7 and 8. Region 5 and 3 are located before the U-bend. The flow is here fully developed and the streamwise component in the cross section shows a typical inverted C shape, with larger velocities near the left wall. This combined with the plot of the cross-flow component (y-velocity, cfn, Fig. 2) shows the presence of two fairly symmetric (with respect to the channel mid-height) longitudinal vortices. On the other hand, Fig. 8 shows the same cross-section after the U-bend, and in this case the vortical structures are not yet formed and the flow is substantially affected by the bend. In this section the difference between experiments and simulations appears to be larger, suggesting a limitation in modeling turbulence in the bend region.

Finally, the error in the computed velocities is reported in Fig. 9 for a vertical section before the U-bend. The CFD results are interpolated on the experimental locations and the differences in the three velocities components (normalized by the experimental values) are plotted. The largest error is about 5% and, most importantly, has a unorganized pattern, showing that the same flow features (streamwise vortices and secondary rib structures) are present.

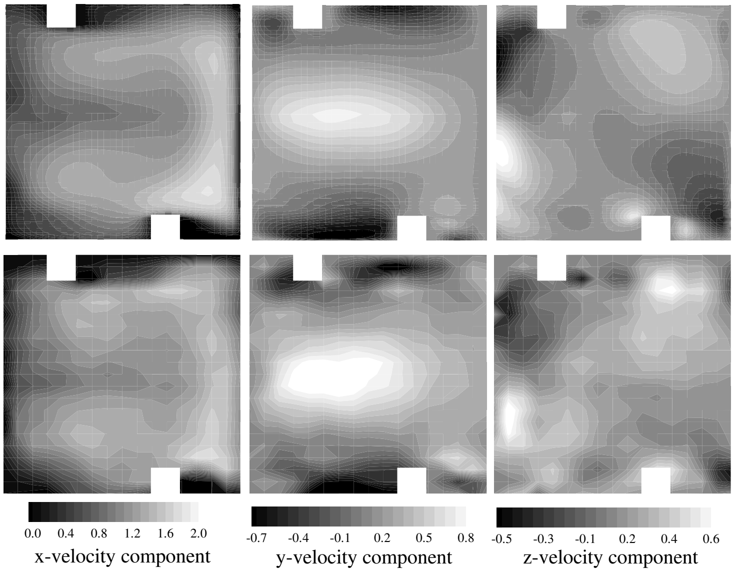

FIGURE 7. Comparison between simulations (top) and experiments (bottom) in region 3 (see Fig. 2): vertical plane corresponding to a cross-section before the U-bend.

Results obtained on coarse grids show the same main features and agree fairly well with the experiments.

6. Conclusions and future work

A numerical and experimental investigation of the turbulent flow inside a multiple-leg serpentine with oblique ribs is presented. Both the simulations and the measurements are based on the use of novel techniques: the immersed boundary approach for the numerical predictions and the magnetic resonance velocimetry for the experiments. Both technique are very fast. Once the experimental stand is setup, large amount of data can be measured in very short time for a CAD model that fits into the measuring section. The same CAD model is used to set up the computations without extra cost in time.

Extremely detailed comparisons are reported due to the high resolution of the two approaches considered: up to 40 million grid cells are used in the simulations and about 2 million point measurements are available. The results show an overall satisfactory agreement and the turbulence model used (two-equation model) seems to perform quite well for the straight parts of the passage capturing the three-dimensional flow structures accurately. The major discrepancies are present in the U-bend and in the immediate downstream areas where the linear eddy viscosity approaches used cannot account for the strong turbulence anisotropy (Iacovides & Raisee 1999).

Additional simulations and experiments have been carried out at a larger Reynolds number and the results show again an encouraging agreement. Future work will be devoted to perform similar comparisons between these two techniques in different complex configurations; in particular, in the cardiovascular system.

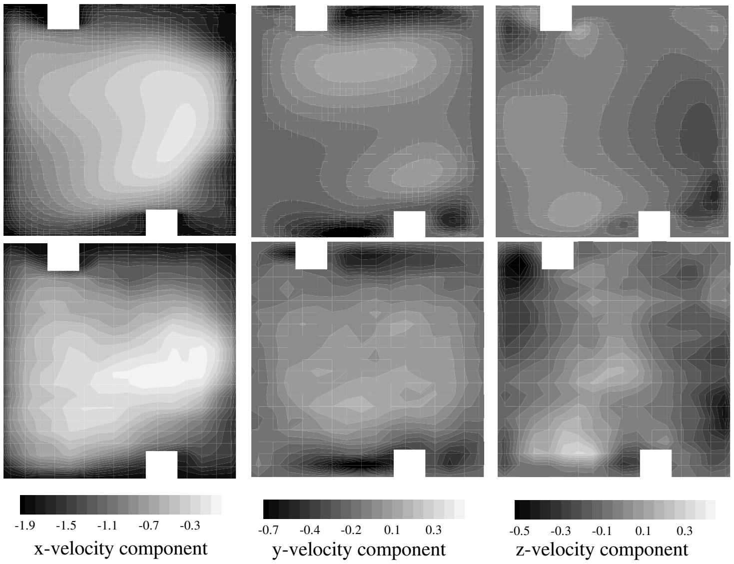

FIGURE 8. Comparison between simulations (top) and experiments (bottom) in region 4 (see Fig. 2): vertical plane corresponding to a cross-section after the U-bend.

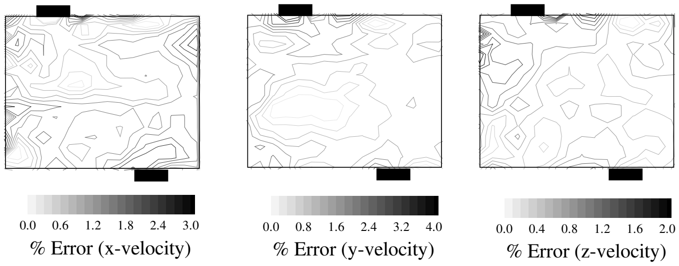

FIGURE 9. Differences between computations and experiments in region 3 (see Fig. 2): vertical plane corresponding to a cross-section before the U-bend.

Acknowledgements

The authors gratefully acknowledge Mr. Chuck Booten of Stanford University for designing the internal passage and Prof. Ryan Wicker and Mr. Francisco Medina at the Unversity of Texas at El Paso for manufacturing it.

REFERENCES

CUMPSTY, N., 1997 *Jet propulsion.* Cambridge University Press, Cambridge UK.

ELKINS, C. S., MARKL, M., PELC, N. & EATON, J.K., 2003 4D magnetic resonance velocimetry for mean flow measurements in complex turbulent flows. *Exp. Fluids* **34** 494–503.

Fukushima, E. 1999 Nuclear magnetic resonance as a tool to study flow. *Ann. Rev. Fluid. Mech.* **31** 95–123.

Han, J. C., Park, J. S. & Lei, C. K., 1985 Heat transfer enhancement in channels with turbulence promoters. *J. of Engr. for Gas Turbines and Power*, **107**, 628-635.

Kalitzin, G. & Iaccarino, G., 2002 Turbulence modeling in an immersed boundary RANS method. *Annual Research Briefs*, Center for Turbulence Research, 415–426.

Iaccarino, G. & Verzicco, R., 2003 Immersed boundary technique for turbulent flow simulations. *Appl. Mech. Rev.* **56** 331–347.

Iaccarino, G., Ooi, A., Durbin, P. A. & Behnia, M., 2002 Conjugate heat transfer predictions in two-dimensional ribbed passages. *Int. J. Heat and Fluid Flow*, **23**, 340–346.

Iacovides, H. & Raisee, M., 1999 Recent progress in the computation of flow and heat transfer in internal cooling passages of turbine blades. *Int. J. Heat and Fluid Flow*, **20**, 320-328.

Ooi, A., Iaccarino, G., Durbin, P. A. & Behnia, M., 2002 Reynolds averaged simulations of flow and heat transfer in ribbed ducts. *Int. J. Heat and Fluid Flow*, **23**, 750–757.

Ooi, A., Petterson Reif, B. A., Iaccarino, G. & Durbin, P. A. 2002 RANS calculations of secondary flow structures in ribbed ducts. *Proc. of the 2002 Summer Program*, Center for Turbulence Research, 43–53.

Rau, G., Cakan, M., Moeller, D. & Arts, T., 1998 The effect of Periodic ribs on the local aerodynamic and heat transfer performance of a straight cooling channel. *J. Turbomachinery*, **120**, 368-375.

Spalart, P. R. & Allmaras S. R., 1994 A one-equation turbulence model for aerodynamic flows. *La Recherche Aerospatiale*, **1**, 1–23.