12TH ERCOFTAC/IAHR/COST WORKSHOP ON REFINED TURBULENCE MODELLING

OCTOBER 12-13, 2006, TECHNICAL UNIVERSITY OF BERLIN, GERMANY

F. Thiele[1] *and S. Jakirlić*[2]

[1]*Institute of Fluid Mechanics and Technical Acoustics*
Technical University of Berlin, Müller-Breslau-Str. 8, 10623 Berlin, Germany

[2]*Institute of Fluid Mechanics and Aerodynamics*
Darmstadt University of Technology, Petersenstr. 30, 64287 Darmstadt, Germany

Abstract

A short report is given of the 12th ERCOFTAC SIG 15 Workshop on Refined Turbulence Modelling, which was jointly organized with the IAHR Working Group for Refined Flow Modelling. Also the COST action has been involved in co-sponsoring some of the workshop activities.

1 Introduction

The role of the ERCOFTAC SIG15 (Special Interesting Group for Turbulence Modelling)* series of workshops on refined turbulence modelling is closely connected to intensive verification and systematic validation of CFD (Computational Fluid Dynamics) technology for solving the problems of both fundamental importance and industrial relevance. Focus is on the credibility and reliability of both the numerical methods and mathematical models simulating turbulence. In such a way a large database of simulation results along with detailed comparison with the reliable reference data (experimental, DNS and highly-resolved LES databases) has been assembled. The SIG15 workshops promote the discussion and conclusions about predictive performance of variety of statistical turbulence models, SGS models in the LES-framework as well as hybrid LES/RANS models in a broad range of well-documented flow configurations under the scientists, researchers, users and developers from industry and from the academic field.

The 12th ERCOFTAC/IAHR/COST Workshop was held on 12th and 13th October, 2006 at the Technical University of Berlin, Germany. Similar to the previous eleven workshops in Lyon (1991), Manchester (1993), Lisbon (1994), Karlsruhe (1995), Chatou (1996), Delft (1997), Manchester (1998), Helsinki (1999), Darmstadt (2001), Poitiers (2002) and Gothenburg (2005) some fundamental phenomena, but also some industrially relevant problems have been chosen as test cases for this workshop. The selection of test cases was made by the steering committee of the SIG 15. The following four flows involving numerous features of scientific and engineering relevance (complex geometry; unsteady, (nominally) 2-D and 3-D separation and reattachment, vortex sheets, swirling effects, etc.) were finally selected as test cases for this workshop:

Case 11.1: Flow over a wall-mounted, 2-D hump with oscillatory zero-mass-flux jet or suction through a thin slot (Exp.: Greenblat et al., 2004, 2005)

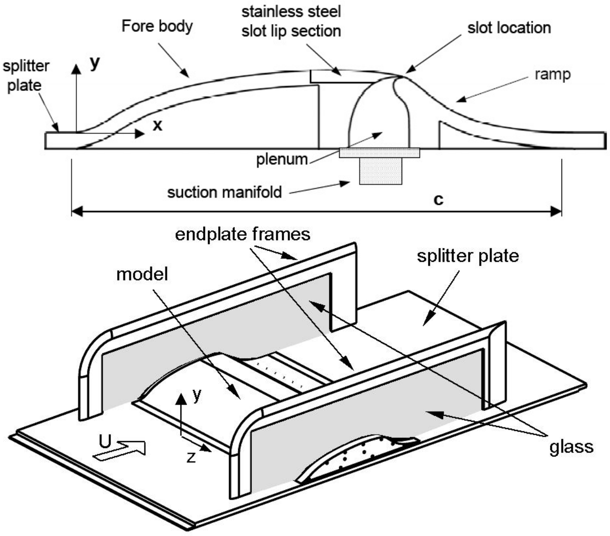

Figure 1: Cross-section schematic through the model and isometric view showing the coordinate system used (adopted from Grenblat et al., 2004)

Case 11.2: Flow over a symmetric 3-D hill (Exp.: Byun et al., 2002; Simpson et al., 2003)

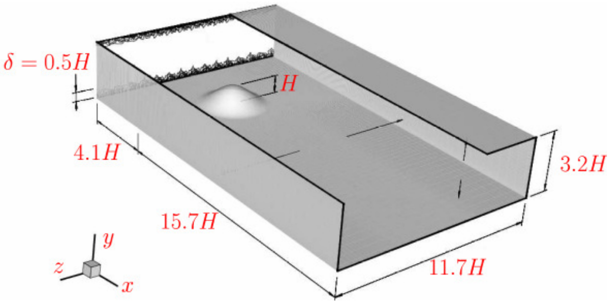

Figure 2: Solution domain schematic (Davidson, 2005)

---

[*] The steering committee members are K. Hanjalic, S. Jakirlic, D. Laurence, B.E. Launder, M.A. Leschziner, R. Manceau, F. Menter, W. Rodi and S. Walin.

Case 12.1: Tip-gap turbulent flow in a low-speed, linear compressor cascade (Exp.: Tian et al., 2004, Tang, 2004)

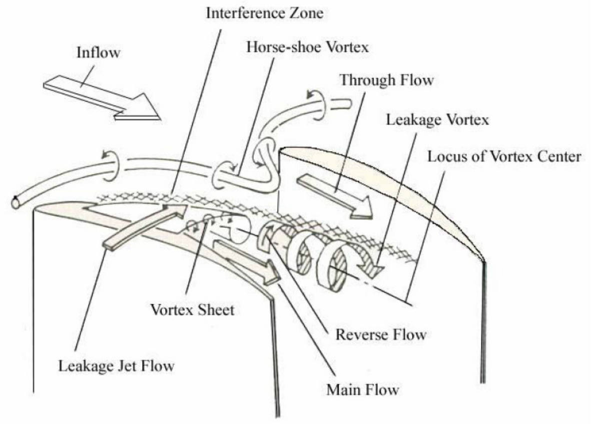

Figure 3: Flow features in the blade tip region (adopted from Inoue and Kuroumaru, 1988)

Case 12.2: Flow and mixing in a model of swirl combustor (Exp.: Palm et al., 2005, 2006)

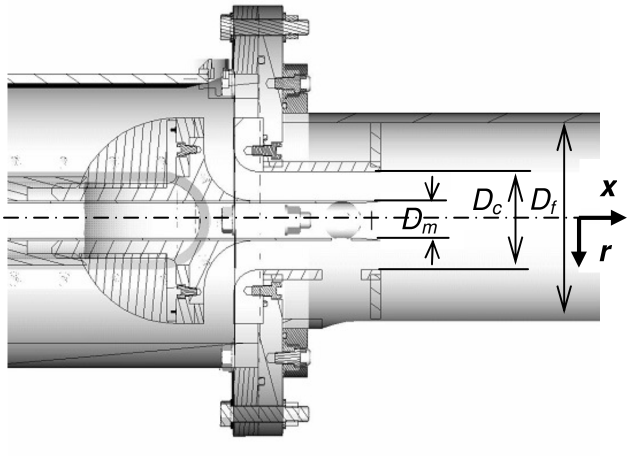

Figure 4: Combustion chamber model

The two first test cases, 11.1 and 11.2, have already been selected for the preceding workshop held at the Chalmers University in Gothenburg in April 7-8, 2005.

The 12th Workshop was attended by 24 participants from Europe and U.S.A. (17 from Germany, 1 from France, 2 from Great Britain, 1 from Denmark, 1 from Sweden, 1 from Russia and 1 from U.S.A.): of which 4 from industry (Rolls-Royce; ANSYS GmbH; Oil & Gas Machinery Dynamics, Copenhagen), 2 from research institutes (DLR, FOI) and 18 from universities.

2 Short Summary of Results and Discussion

Flow description, instructions for calculations, detailed specification of the shape and dimensions of solution domains, as well as of the inlet data and boundary conditions for all four test cases considered, are given in the workshop proceedings[†]. Here, only a short description of all four test cases and a summary of some specific outcomes and the most important conclusions are given.

Case 11.1: Flow over a wall-mounted, 2-D hump

This is a carry-over test case from the recent workshops (NASA Langley, March 2004, Rumsey et al., and 11th ERCOFTAC/IAHR Workshop, April 2005, Gothenburg, Sweden). Experimental data are available for the three cases: baseline configuration (no control), suction control case (steady suction rate through the slot of *0.01518 kg/s*) and zero-efflux oscillatory forcing trough the slot, with nominal peak velocity of *26.6 m/s* and frequency of *138.5 Hz*. The configuration is nominally two-dimensional, although the end plates bring some 3D effects. The hump is *420 mm* long with the crest of *53.7 mm* and is mounted on a splitter plate of thickness *12.7 mm*, which extends *1935 mm* upstream from the hump leading edge and *1129 mm* downstream from the hump leading edge. The hump with the splitter plate is placed in a wind tunnel of *771 mm* width and *508 mm* height, but the nominal test section height (between the splitter plate and the top wall) is *382 mm* and the nominal hump width (between the two end plates) is *584 mm* (Fig. 1). The characteristic Reynolds number based on the hump length is about 10^6 and the Mach number is 0.1. Results containing base plate pressure and friction factor, and PIV of mean *U* and *V* velocity, *uu*, *vv* and *uv* stress components are available at different stations for all three cases. Detailed description of the test case can also be found on *http://cfdval2004.larc.nasa.gov/case3.html.*

The oncoming flow is characterized by a zero-pressure-gradient turbulent boundary layer, whose thickness δ is approximately *57%* of the maximum hump height (h_{max}*=53.74 mm*) measured at the location about two chord lengths upstream of the hump leading edge (coinciding with the origin of the coordinate system, Fig.1), corresponding to the momentum-thickness-based Reynolds number $Re_\theta=7500$. The latter result was obtained by applying a near-wall, second-moment closure model, Saric et al. (2006). Only the profiles of the mean velocity and streamwise stress component are available from the reference experiment.

The four computational groups:

- TUD – *Technical University of Darmstadt*, Germany (S. Saric, B. Kniesner, P. Altenhöfer and S. Jakirlic): LES, DES and a zonal hybrid LES/RANS (HLR) method (Jakirlic et al., 2006) were used,
- UP – *University of Poitiers*, France (R. Manceau): Elliptic-Blending Second-Moment Closure (EB-SMC) model (Manceau, 2005) was used,
- NASA *Langley Research Centre* (C. Rumsey and T. Gatski): Spalart-Allmaras (SA), k-ω SST and EASM (Explicit Algebraic Reynolds Stress model

---

[†] *http://www.cfd.tu-berlin.de/ercoftac-workshop06.html* or *http://www.ercoftac.org*

in conjunction with ω-equation) model were used and

- TUB – *Technical University of Berlin* (C. Mockett, A. Carnarius and F. Thiele): different versions of the SA model and the linear and non-linear, ω-equation-based models were used

contributed to the comparative, cross-plot analysis of all three cases. Here only the results with respect to the main separation and reattachment locations are displayed, Figs. 5.

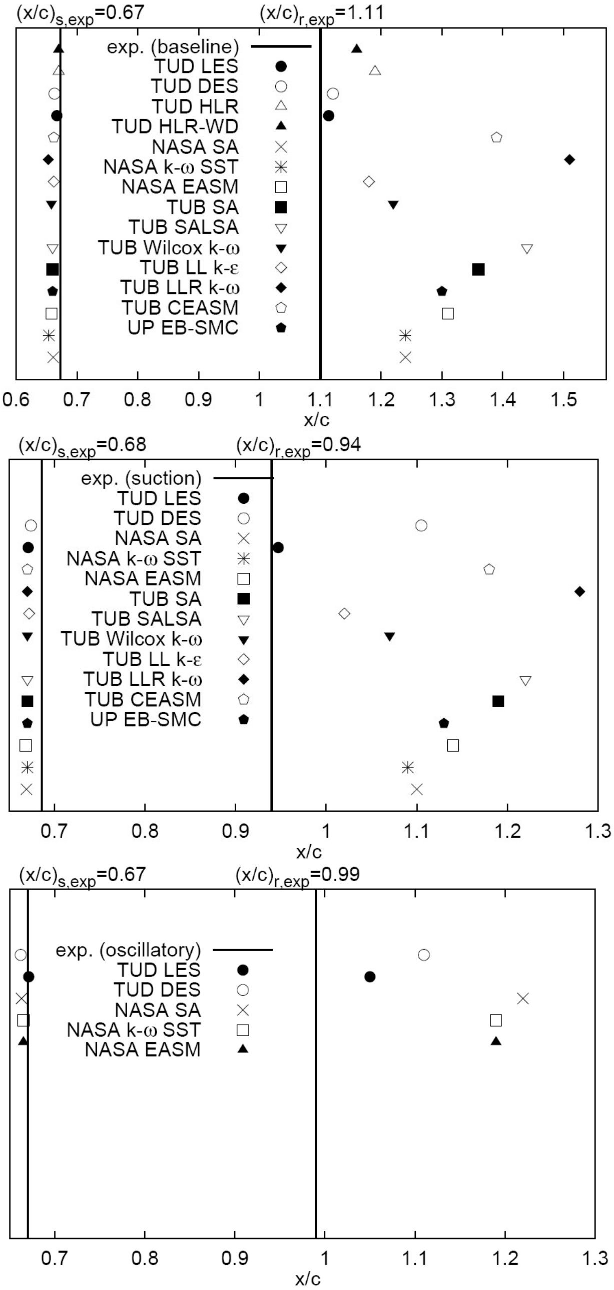

Figure 5: The computationally obtained separation and reattachment locations compared to the experimental results for all three cases investigated: baseline configuration (upper), steady suction configuration (middle) and oscillatory blowing/suction configuration (lower)

The effect of the boundary layer forcing on the recirculation zone shortening, with the steady suction representing the most effective flow control mode, can be clearly recognized. Both the LES and DES (apart from the steady suction case; here, the weakness of the original DES method, Spalart et al. (1997), with respect to the in-advance-determined, flow-independent interface position came into play) results are in closest agreement with the reference experiment with respect to both separation and reattachment locations. This is not valid for the various RANS models applied. All RANS models reproduced a fairly weak gradient of the shear stress components at this location (not shown here), as a consequence of a generally low shear-stress level in the shear layer being aligned with the mean dividing streamline, causes a longer recirculation region. The latter is a typical outcome of the RANS method, almost independent of the modelling level adopted. The investigated flow configuration is characterized by unsteady separation governed by large-scale unsteadiness (highly intermittent separation and reattachment regions, highly unsteady separated shear layer), all the features being beyond the reach of the inherently steady RANS approach. A detailed cross-plot presentation of all results obtained can be downloaded from the workshop's web site. Some further computational results can be found in the works of Krishnan et al. (2004) and Saric et al. (2006).

This was the only case which was computed by more groups. The other test cases were each handled by only one group (the only exception is the case 11.2, which was simulated by two groups), allowing no cross-plot analysis. These results were displayed and discussed in the framework of four individual presentations.

Case 11.2: Flow over a symmetric 3-D hill

The experimental arrangement is shown in Fig. 6. The hill, of height *H=0.078* m, is located in a duct of height *3.205H* and width *11.67H*.

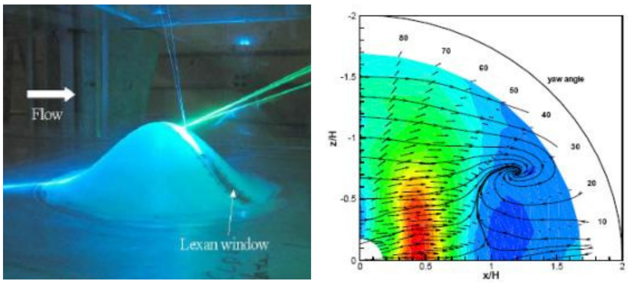

Figure 6: Experimental configuration and topology on rear portion of surface

The hill is subjected to a boundary layer of thickness *0.5H*, this value prevailing at *2H* upstream of the hill crest, created in the upstream duct with the aid of roughness elements of the kind used to represent an atmospheric boundary layer in a wind tunnel. The boundary layer on the upper wall is very similar to that on the lower wall. While this does not interact directly with the hill, it will affect it by potential mechanisms. The Reynolds number, based on the maximum inlet velocity ($U_{ref} = 27.5\ m/s$ in the experiment) and the hill height *H*, is $Re = 1.3x10^5$. The turbulence intensity outside the boundary layer is reported be about *0.1%*. Pressure, LDA and some HWA data have been obtained by Simpson et al. (2002) and Byun et al. (2003). Field measurements are available in the form of profiles of the mean-flow velocities and Reynolds stress components at *3.63* hill-heights downstream of the hill crest. In addition, hill-topology results, surface pressure, the velocity field in the hill centre-plane and some near-surface velocity and turbulence data are reported.

This case was simulated by two numerical groups

- *Imperial College London*, UK (Li, N. and Leschziner, M.A.): simulations were undertaken with meshes containing between 1.5 and 9.6 million nodes, the finest-grid large-eddy simulation approaching full wall resolution. Results from a fully wall-resolving 36.7 million-node large-eddy simulation were also presented. Coarser grids were used in conjunction with the zonal scheme, wherein the interface was placed within y^+=20-40 and 40-60, using 3.5 and 1.5 million nodes, respectively. The inlet boundary layer, at -4H, was generated by a combination of RANS and LES precursor calculations, the former matching the experimental mean-flow data and the latter providing the spectral content.
- *University of Karlsruhe*, Germany (Garcia-Villalba, M. and Rodi, W.): large-eddy simulation using dynamic Smagorinsky model was performed on a grid containing in total 153.5 million grid cells (19 million cells were used for a precursor simulation for inflow generation; body force technique developed by Pierce (2001) was applied)

The three-dimensional separation pattern being characterised by multiple vortical structures in the leeward side of the hill is far more complicated than in the previous test case. Wang et al. (2004) show that RANS models, at whatever level of sophistication, similar to the findings in the 2-D, wall-mounted hump, seriously over-estimate the size of the separation bubble and flow intensity in this region. The readers interested in more details about the results obtained by LES, different non-linear eddy-viscosity and second-moment closure models and an approximate ‘zonal’ near-wall treatment applied within a LES strategy are referred to the works of Wang et al. (2004) and Tessicini et al. (2007).

Case 12.1: Tip-gap flow in a compressor cascade

The related experimental studies have been conducted in a linear compressor cascade in a low speed wind tunnel in the department of aerospace and ocean engineering at Virginia Tech, Blacksburg with special emphasis on the tip leakage flow development near and in the tip clearance. The flow characteristics are illustrated in Fig. 3. The tip gap flow is highly skewed three-dimensional flow throughout the full gap. The tip gap flow interacts with the primary flow, separates from the endwall, and rolls up on the suction side to form the tip leakage vortex. The tip leakage vortex produces high turbulence intensities. The tip gap flow correlations of streamwise and wall normal velocity fluctuations decrease significantly from the leading edge to the trailing edge of the blade due to flow skewing contributing to a strongly anisotropic turbulent flow.

Details of the aerodynamic design of the cascade tunnel are given in the work of Tian et al. (2004). There are two three-quarter inch high suction slots on the upper and lower endwall at 7.48 inch in front of the cascades and with these slots, the inlet flow boundary layer is removed, Fig. 7-lower. The flow is tripped by a square bar mounted on the lower suction slot as shown in Fig. 7-lower. Boundary layer trips (sand paper) are also used on both side of surface of the blades. The optic glass insert has an eight inches diameter. The blades have a chord of 10 inches and an effective span about 10 inches. The blades spacing is 9.29 inches. The blades were instrumented with pressure taps on the surfaces at the mid-span of the blades (#4 and #5) as shown in Fig. 7-upper.

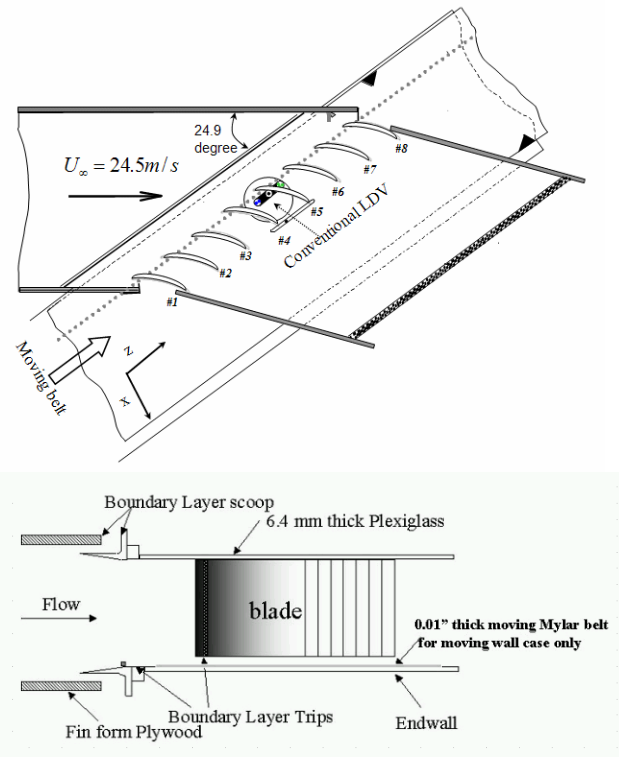

Figure 7: Test section of the linear cascade tunnel (upper) and its side view (lower)

Inserts with different optical glass access positions make LDV measurement under the blade for multiple different positions possible. The nominal running conditions are the speed of *24.5 m/s* and temperature of *25C° ± 1C°*. The bed coordinate system (X_{bed}, Z_{bed}) is aligned with the suction slot and the direction of travel of the belt. The origin of the bed coordinate is at the midpoint between Blade 4 and Blade 5 along with the leading edge of the blade row on the tunnel floor. For the workshop, only the case with *1.65%* tip gap height and with non-moving belt was proposed.

Only the ANSYS GmbH Germany/NTS St.-Petersburg computational group (F. Menter/S. Yakubov) contributed to this flow configuration. Eight solution sets were obtained on three different grids containing 0.9, 1.1 and 3.7 million nodes by using the ANSYS CFX code. Different versions of the k-ω SST model modified to account for the curvature and reattachment corrections and an explicit algebraic Reynolds stress model were applied. The conclusions arising from this study can be summarized as follows:

- Both vortices – tip leakage vortex and tip separation vortex found in the experiments were identified in the RANS solution, but the latter was significantly larger than in the experiment
- In spite of the fact that blade loading and lower wall static pressure distribution were predicted rather accurately, computed positions of the vortex centers and separation line are shifted towards the suction side of the blade, the deviation between the CFD solutions and the experimental data is getting large while moving downstream the passage
- The use of various turbulence models being potentially capable to improve prediction quality of tip-gap flows for this particular test case didn't show any significant difference comparing to the original SST model

The interested readers are encouraged to contact the case coordinator Dr. F. Menter (florian.menter@ansys.com) for more details.

Case 12.2: Flow in a tubo-annular swirl combustor

Flow in an axisymmetric model of a tubo-annular combustor with an axial, non-swirling stream (representing fuel) and an annular swirling jet (representing primary air) expanding into a flue has been experimentally investigated in a range of swirl numbers $0.0 < S < 1.2$ and mass flow rates corresponding to the Reynolds numbers $23500 < Re < 102000$ (central stream) and $49530 < Re < 125500$ (annular flow). In addition to the PIV measurements of the flow in the flue (Palm et al., 2005), the profiles of all variables (mean velocities and Reynolds stresses) at a cross-section (x = -40 mm) within the inlet pipes have been measured by using the LDA technique (Palm et al., 2006). The combustor dimensions are displayed in Fig. 4.

The effects of the increasing swirl intensity on the interaction between the outer, swirling stream and the inner, non-swirling flow in the near field of a model combustor is computationally investigated by the TU Darmstadt group (Saric, Kniesner, Altenhöfer and Jakirlic) applying both RANS (using the eddy-viscosity-based ζ–f model of turbulence, Hanjalic et al., 2004) and LES methods. Both the in-house code FASTEST and the commercial code FIRE (AVL List GmbH, Graz) were used. The increasingly swirled annular jet promotes an intensive mixing in the near field of combustor. It is manifested through the enhanced spreading of the flow into the radial direction and the consequent strengthening of the back-flow activity in the combustor core. The overall agreement between simulations and measurements is good. This is particularly the case in the shear layer and the outer, wall-affected flow region. Some important departures from the experimental results with respect to the mean flow structure are present in the flow core. The simulations return a ring-shaped recirculation zone with positive centreline velocities along entire flue geometry, in contrast to a closed, free separation region detected experimentally. A cause of this deviation lays most probably in the imposed outlet boundary conditions representing inadequately the structure of the combustor outflow. Further work on this issue is necessary. For more details about the computational results the works of Saric et al. (2007) should be consulted.

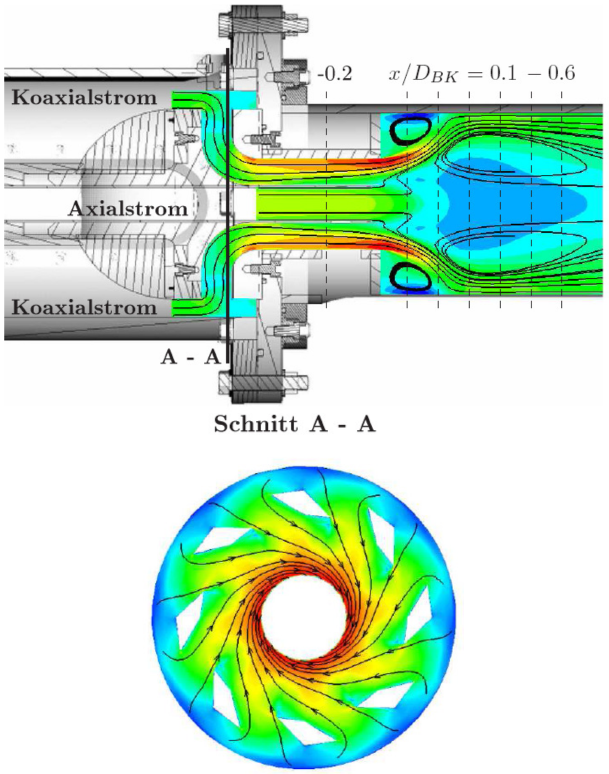

Figure 8: Topology of the flow within the inlet section (including swirl generator) and the flue

3 Conclusions

Short summaries of the test cases description and most important conclusions arising from the computationally obtained body of data are given in this report. Only seven computational groups (five of them are headed by the SIG 15 steering committee members) contributed to the four test cases. This represents the weakest attendance since the establishment of the ERCOFTAC SIG15 workshop series in 1991. There are obvious reasons for such an outcome: the cases proposed are all complex 3-dimensional, unsteady flow configurations whose correct capturing requires the employment of more sophisticated computational methods (e.g., LES, hybrid LES/RANS); accordingly, several months (if not years) of intensive work, which could be done only in the framework of a funded project, are necessary.

Possible remedies to overcome the present situation can be the introduction of some geometrically simpler benchmarks but featured by complex flow and turbulence phenomena as well as the initiation of some survey lectures instituting appropriate sessions. Furthermore, a work on establishment of the SIG-15 Forum (members from industry, research centres, CFD companies and academia) should be undertaken in order to establish a kind of advisory board which will be actively involved in the life of the SIG15 (e.g. test case selection in a market-search-way, enabling the potential contributors would be known in advance). The potential members, also the colleagues from some non-European countries (e.g., China, Japan, USA), have to be recognized and directly contacted.

Acknowledgements

We would like to thank to the large number of people who were involved in the preparation and execution of this workshop. This applies especially to the SIG15 Steering Committee and the staff at the Institute of Fluid Mechanics and Technical Acoustics in Berlin (D. Eschricht and C. Mockett) and Chair of Fluid Mechanics and Aerodynamics in Darmstadt (S. Saric and B. Kniesner), who in one way or the other were all involved in the organization. We are thankful to the ERCOFTAC Administration/Development Office for the student grants. The contribution of the reference data suppliers is gratefully acknowledged.

References

[1] Byun, G., Simpson, R.L. and Long, C.H. (2003): A Study of Vortical Separation from Three-Dimensional Symmetric Bumps, AIAA-2003-0641

[2] Davidson, L. (2005): 11th ERCOFTAC SIG15 Workshop on Refined Turbulence Modelling, Chalmers University, Gothenburg, April 7-8

[3] Greenblatt, D., Paschal, K.B., Yao, C.S., Harris, J., Schaeffler, N.W. and Washburn, A.E. (2004): A separation control CFD validation test case, Part1: baseline and steady suction, AIAA Paper No. 2004-2220

[4] Greenblat, D. Paschal, K.B., Yao, C.S. and Harris, J. (2005): A separation control CFD validation test case. Part 2: Zero Efflux oscillatory blowing, AIAA Paper No. 2005-0485

[5] Hanjalic, K., Popovac, M. and Hadziabdic, M. (2004): A robust near-wall elliptic-relaxation eddy-viscosity turbulence model for CFD. Int. J. Heat Fluid Flow, **25**, pp. 1047-1051

[6] Inoue, M. and Kuroumaru, M. (1988): Structure of a Tip Clearance Flow in an Isolated Axial Compressor Rotor, ASME Paper 88-GT-251

[7] Jakirlić, S., Kniesner, B., Šarić S. and Hanjalić, K. (2006): Merging near-wall RANS models with LES for separating and reattaching flows, ASME Paper No. FEDSM2006-98039

[8] Krishnan, V., Squires, K.D. and Forsythe, J.R. (2004): Prediction of separated flow characteristics over a hump using RANS and DES. AIAA Paper No. 2004-2224

[9] Manceau, R. (2005): An improved version of the elliptic blending model: application to non-rotating and rotating channel flows. *4th Int. Symp. on Turbulence and Shear Flow Phenomena*, Williamsburg, VA, USA, June 27-29

[10] Palm, R., Grundmann, S., Jakirlić, S. and Tropea, C. (2005): Experimental investigation and modelling of flow and turbulence in a swirl combustor. *4th Int. Symp. on Turbulence and Shear Flow Phenomena*, Williamsburg, VA, USA, June 27-29

[11] Palm, R., Grundmann, S., Weismüller, M., Šarić, S., Jakirlić and Tropea, C. (2006): Experimental characterization and modelling of inflow conditions for a gas turbine swirl combustor. *Int. J. of Heat and Fluid Flow*, **27**, pp. 924-936

[12] Rumsey, C.L., Gatski, T.B., Sellers III, W.L., Vatsa, V.N. and Viken, S.A. (2004): Summary of the 2004 CFD Validation Workshop on Synthetic Jets and Turbulent Separation Control, AIAA Paper No. 2004-2217

[13] Šarić S., , Kniesner, B., Altenhöfer, P., Jakirlić, S., Tropea, C., Čavar, D. and Basara, B. (2007): Swirl intensity influence on interaction between non-swirling and swirling co-axial jets in a combustor configuration: LES and modelling study. *5th Int. Symp. on Turbulence and Shear Flow Phenomena*, Munich, Germany, August 27-29

[14] Šarić, S., Jakirlić, S., Djugum, A. and Tropea, C. (2006): Computational analysis of locally forced flow over a wall-mounted hump at high-Re number. *Int. J. Heat and Fluid Flow*, Vol. 27, pp. 707-720

[15] Šarić, S., Jakirlić, S., Čavar, D., Kniesner, B., Altenhöfer P. and Tropea, C. (2007): Computational study of mean flow and turbulence structure in inflow system of a swirl combustor. *Notes on Numerical Fluid Mechanics and Multidisciplinary Design*, Vol. **96**, Tropea et al. (Eds.), pp. 462-470, Springer Verlag

[16] Simpson, R.L., Long, C.H. and Byun, G. (2002): Study of Vortical Separation from an Axisymmetric Hill, *Int. J. Heat and Fluid Flow,* **23**, pp. 582-591

[17] Spalart, P.R., Jou, W.-H., Strelets, M. and Allmaras, S.R. (1997): Comments on the feasibility of LES for wings and on hybrid RANS/LES approach. Proc. 1st AFOSR Int. Conf. on DNS and LES, Ruston, LA, USA

[18] Tessicini, F., Li, N. and Leschziner. M.A. (2007): Large eddy simulation of three-dimensional flow around a hill-shaped obstruction with a zonal near-wall approximation. *Int. J. Heat Fluid Flow*, **28**, pp. 894-908

[19] Tang, G. (2004): Measurements of the tip-gap turbulent flow structure in a low-speed compressor cascade. PhD Thesis, Virginia Tech, Blacksburg, USA (Supervisor: R. L. Simpson)

[20] Tian, Q., Simpson, R.L. and Tang, G. (2004): Flow visualization on the linear compressor cascade

endwall using oil flows and laser Doppler anemometry. *Meas. Sci. Technol.*, **15**, pp. 1910–1916

[21] Wang C., Jang Y.J. and Leschziner M.A. (2004): Modelling two- and three-dimensional separation from curved surfaces with anisotropy-resolving turbulence closures. *Int. J. Heat and Fluid Flow*, **25**, pp. 499-512