

Numerical simulation of turbulent heat transfer past a backward-facing step: 2D/3D RANS versus IDDES solutions

E M Smirnov¹, A A Smirnovsky¹, N A Schur¹, D K Zaitsev¹, P E Smirnov²

¹ St.-Petersburg Polytechnic University, St.-Petersburg, Russia

² ANSYS Germany GmbH, Otterfing, Germany

E-mail: aero@phmf.spbstu.ru

Abstract. The contribution covers results of numerical study of air flow and heat transfer past a backward-facing step at the Reynolds number of 28,000. The numerical simulation was carried out under conditions of the experiments of Vogel&Eaton (1985), where nominally 2D fluid dynamics and heat transfer in a channel with expansion ratio of 1.25 was investigated. Two approaches were used for turbulence modelling. First, the Menter SST turbulence model was used to perform refined 2D and 3D RANS steady-state computations. The 3D analysis was undertaken to evaluate effects of boundary layers developing on the sidewalls of the experimental channel. Then, 3D time-dependent computations were carried out using the vortex-resolving IDDES method and applying the spanwise-periodicity conditions. Comparative computations were performed using an in-house finite-volume code SINF/Flag-S and the ANSYS Fluent. The codes produced practically identical RANS solutions, showing in particular a difference of 4% in the central-line peak Stanton number calculated in 2D and 3D cases. The IDDES results obtained with two codes are in a satisfactory agreement. Comparing with the experimental data, the IDDES produces the best agreement for the wall friction, whereas the RANS solutions show superiority in predictions of the local Stanton number distribution.

1. Introduction

Turbulent flow past a backward-facing step plays an important role in benchmarking the performance of turbulence models for separated flows with and without heat transfer. Separation and reattachment peculiar to this flow cause large variations in the wall friction and heat transfer rates. The flow peculiarities cause difficulties in providing accurate prediction of the flow fine structure, dynamic and thermal boundary layers, and resulting skin friction coefficient and heat transfer coefficient.

Among the existing experimental heat transfer studies in separating and reattaching turbulent flows the most known one is that of Vogel and Eaton [1], who performed a combined study of fluid dynamics and heat transfer past a backward-facing step in a channel with expansion ratio of 1.25. The data reported in [1] has added the fluid dynamics results obtained previously by Adams et al [2]. Both the studies were performed in a low-speed, open-circuit wind tunnel with a test section that had the aspect ratio (spanwise-extent/step-height) of 11.4. It was assumed that the flow and heat transfer in the central plane of the test section could be treated as two-dimensional.

Experimental data given in [1,2] for the Reynolds number of 28,000 has been widely used to evaluate the performance of various RANS turbulence models for separating and reattaching flow and heat transfer predictions. It has been established that the k - ε models developed by the early 1990s are generally incapable of an accurate prediction of the fluid dynamics and heat transfer in these types of



flow [3]. A definite success was achieved only with the AKN-version of the $k-\varepsilon$ model [4]. The Wilcox $k-\omega$ model and the Menter SST model (without tuning) have shown [5] a good agreement with the experimental data [1] for the reattachment length and the Stanton number distribution. However, negative values of the skin friction coefficient in the separation zone were overestimated by both the models. These findings were confirmed in particular by a recent contribution [6] where a special attention was paid to get grid independent solutions. A similar behaviour has shown by two versions of the v^2-f turbulence model examined in [7].

A considerable effort was made in application of the Large Eddy Simulation (LES) technique to predict all the flow features and heat transport mechanisms in the flow over a backward-facing step. As applied to the experimental flow conditions [1,2], the LES results reported in [8,9] show that a good agreement is achieved for the Stanton number distribution, but similar to above mentioned RANS predictions negative values of the skin friction coefficient in the separation zone are overestimated.

Progress in increasing accuracy of high Reynolds number separating and reattaching flow computations is expected also in line of developments of hybrid RANS/LES approaches. A great practical potential has the improved version of the Detached Eddy Simulation method developed in [10] and termed IDDES. Among other test cases, validation of the IDDES method reported in [10] included also the backward-facing step flow studied in [1,2] and has shown a good performance of the IDDES for prediction of flow dynamics. Results of the IDDES calculations of heat transfer past a backward-facing step are still absent in the open literature.

All the known numerical studies dealing with the flow conditions adopted in the experiments [1,2] assumed the flow and heat transfer statistically two-dimensional. First part of the present contribution covers RANS-based numerical analysis of 3D effects associated with boundary layers developing on sidewall of the test section. The second part is devoted to IDDES of statistically-2D flow dynamics and heat transfer under the experimental conditions [1,2].

2. Test case description

The experiments [1,2] chosen for simulation were conducted in an open-circuit wind tunnel. The Reynolds number, based on step height, H , and reference free-stream velocity, U_{ref} , was approximately 28,000. The development section upstream of the step was relatively long and the sidewall boundary layers were removed using suction boxes placed $12H$ upstream of the step. The height of the development section was $4H$, giving an expansion ratio of 1.25. The aspect ratio (spanwise-extent/step-height) of the test section was 11.4. For the test case used as reference for the present calculations, the boundary layer thickness at both the step-wall and the upper wall was about $1.07H$ at the position $x=-3.8H$ (momentum thickness was of $0.0123H$). Reference velocity was measured upstream of the step at $x = -3.3H$ in the flow core. Heat transfer measurements were made in air with a Prandtl number of 0.71. On the lower wall, the heat flux, q_w , was 270Wm^{-2} . All other walls were adiabatic. A schematic of the flow configuration is given in Figure 1.

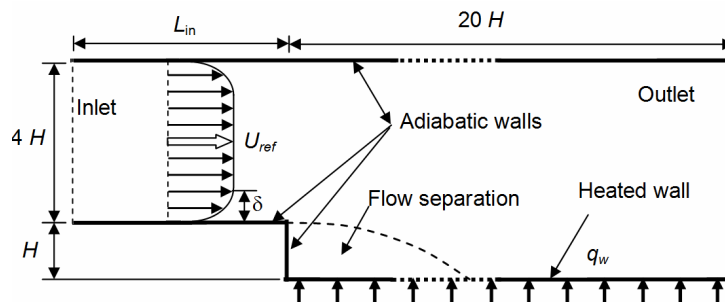


Figure 1. Geometry and flow configuration for the backward-facing step test case calculations.

3. Mathematical models and CFD software

3.1. Governing equations and turbulence models

First series of the present computations was carried out on the base of the Reynolds-averaged Navier-Stokes (RANS) equations closed with the Menter SST turbulence model [11]. 2D and 3D problem formulations were used to get steady-state solutions. The second series was performed using the SST IDDES approach assuming hybrid time-dependent 3D computations on the base of unsteady RANS and LES filtered equations. Actually, a simplified formulation of the SST IDDES suggested in [12] was used for the present computations.

3.2. CFD software used

Comparative computations were performed using an in-house code under development at the St.-Petersburg Polytechnic University and the ANSYS Fluent 14.0. The in-house code termed SINF/Flag-S is based on the unstructured-grid finite volume method of nominally second-order accuracy in space and time. For RANS simulation, the SIMPLEC algorithm was applied when using the in-house code, whereas the pressure-based coupled solver was used running Fluent. For convective terms, the spatial discretization was performed with the QUICK scheme in case of the in-house code and with the second order upwind scheme in case of Fluent.

IDDES simulations were performed with non-iterative time adjustment schemes both running Fluent and the in-house code. The scheme implemented in the in-house code is based on the fractional step method with explicit second-order extrapolation of convective terms in time. The Crank-Nicolson scheme is used for time advancing, and the Rhie-Chow correction is introduced to avoid odd-even spatial oscillations on collocated grids. The spatial discretization is performed with the central scheme for convective terms and with the second-order central scheme for viscous/diffusion terms.

4. RANS-based computations

4.1. Problem definition

Figure 2 shows computational domain used for 3D RANS calculations. The computational domain extended from $-12H$ to $20H$ in streamwise direction ($x = 0$ corresponds to the step location). In the spanwise direction, the size of the domain was $11.4H$. The domain was covered by a grid of 8,060,196 cells with 267 points in the spanwise direction. Fragment of the X-Y plane computational grid is shown in Figure 3. Distributions of grid points over cross sections positioned upstream of the step is illustrated in Figure 4.

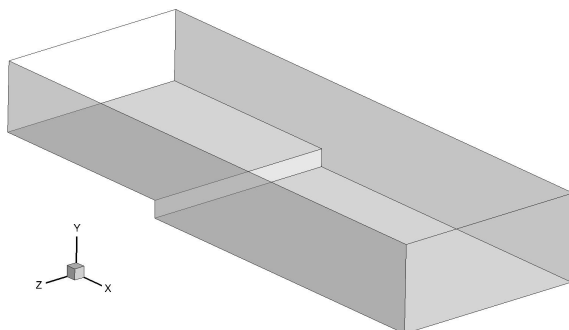


Figure 2. Computational domain for 3D RANS-based calculations.

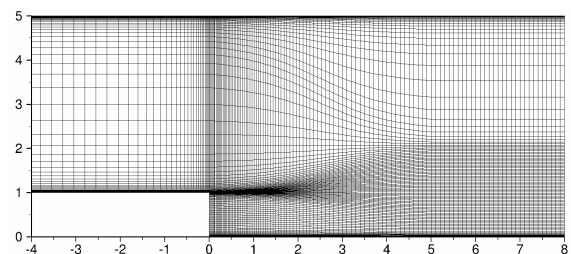


Figure 3. Fragment of the X-Y plane computational grid in the vicinity of the step.

At the domain inlet, Y -distributions of velocity and turbulence quantities were specified, which were obtained from a precursor 2D calculation performed for the channel flow $4H$ width. These inlet distributions were uniform in the spanwise direction and provided experimental value of the boundary layer thickness at the location of $x=-3.8H$. The sidewall boundary layers started to develop from the inlet section as “fully turbulent”. On the outlet boundary a constant pressure was specified. Constant heat flux was specified on the lower wall. The computations were performed at the Reynolds number of 28,000.

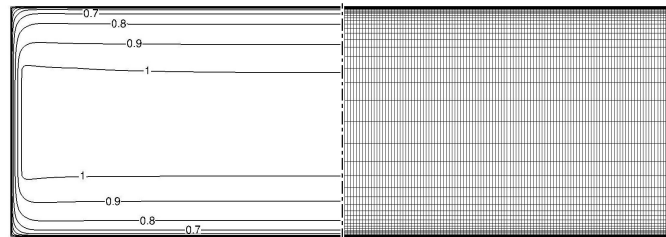


Figure 4. Computational grid and isolines of normalized streamwise velocity at the channel cross-section positioned upstream of the step at $x/H = -1.0$.

In Figure 4 one can see that in the cross-section positioned closely upstream of the step the well resolved boundary layers on side-walls are much thinner as compared with the boundary layers on the step-wall and the upper wall.

4.2. Results of RANS simulations

Before getting 3D solutions, preliminary computations were carried out on the base of the traditional 2D RANS formulation of the problem, using the same grid in the X - Y plane. Computed flow structure past the backward-facing step and the temperature map is illustrated in Figure 5. With the grid used, one is able to resolve multiple-vortex structure of the separation zone including a very small corner vortex near the step.

Streamwise distributions of the skin friction coefficient, C_f , and the local Stanton number, St , calculated with the two CFD codes used are shown in Figure 6. As in the experiments, calculations of C_f and St are based on using the free-stream reference velocity at section $x = -3.3H$. The inlet temperature is taken as a reference one. It is seen that Fluent and the in-house code produce practically identical RANS 2D solutions. It has established that this conclusion is valid for the case of 3D computations as well.

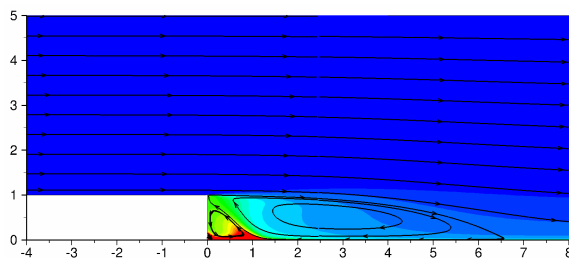


Figure 5. Separated flow structure and temperature field: 2D RANS solution.

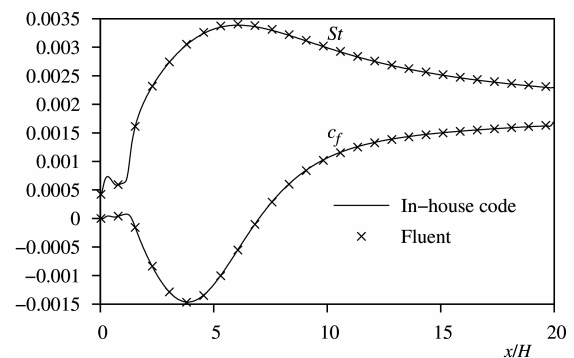


Figure 6. Skin friction coefficient and Stanton number distributions calculated with two CFD codes with 2D RANS problem formulation.

Let us consider the flow structure from the 3D RANS simulation. As one can see in Figures 7 and 8, there are relatively strong side-wall effects in the recirculation bubble adjacent to the step, $0 < x/H < 1.5$, whereas at $x/H > 1.5$ the flow and heat transfer in the major part of the channel is almost two-dimensional. The 3D side-wall effects are attributed to secondary flows arising in the step-sidewall corner regions. The secondary flow leads to increasing heat transfer rate in the corner regions but decreases heat transfer near the central part of the recirculation bubble (Figure 7). The convection effect of the secondary flows is illustrated in Figure 8, where cross-flow streamline pattern and map of normalized temperature, θ , for the cross-section positioned at $x/H = 0.6$ are shown. Temperature is normalized as follows $\theta = (T - T_{in}) / \Delta T_s$, where T_{in} is inlet temperature. Characteristic temperature difference ΔT_s is evaluated as $0.01 q_w \cdot H / \lambda$, where λ is the fluid heat conductivity.

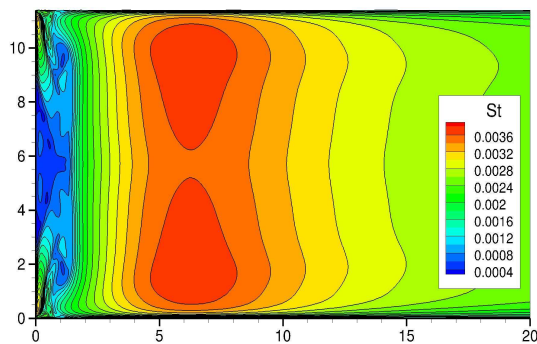


Figure 7. Distribution of the Stanton number over the heated wall obtained with the 3D RANS model.

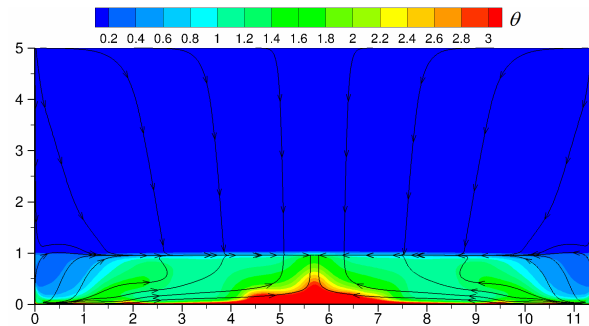


Figure 8. Normalised temperature map and cross-flow streamlines in the cross-section positioned downstream of the step at $x/H = 0.6$

Figure 9 shows that the distributions of Stanton number and skin friction coefficient along the center line of the domain obtained in the 3D RANS case is closed to those computed with the 2D RANS model (Figure 9). The biggest difference (of about 4%) is observed in recirculation zone for the Stanton number, especially near the step. So, one may conclude that despite of the secondary flow existence the flow can be treated as two-dimensional at least for the wall characteristics evaluation. The computed Stanton number distributions are in a good agreement with the experimental data [1]. However, similar to the results of previous works [5-7], negative values of the skin friction coefficient in the separation zone are overestimated as compared with the reference measurements [2].

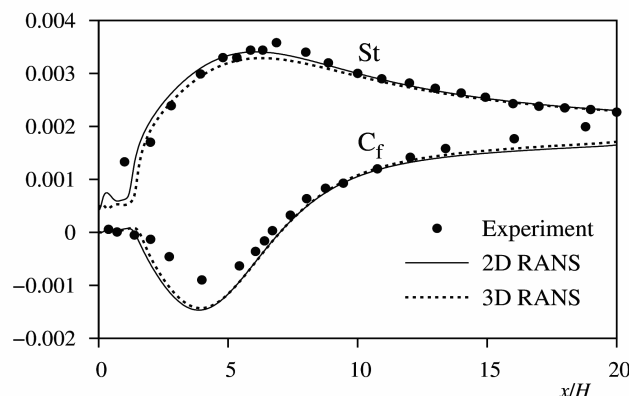


Figure 9. Comparison of the skin friction and Stanton number experimental distributions with those calculated with 2D and 3D RANS formulations.

5. IDDES simulation

5.1. Problem definition

The 3D RANS analysis presented above gives an additional evidence that the flow and heat transfer processes in the test section of experiments [1,2] may be treated as statistically two-dimensional. Consequently, IDDES simulations were carried out prescribing periodic conditions in the spanwise direction. Computational domain with the following parameters was used: inlet boundary was located at $x = -3.8H$, the domain width in spanwise direction was equal to $4H$. Other geometric parameters were the same as in the RANS case. The computational grid shown in Figure 10 contains about 2.2 million cells; the cell size in spanwise direction is uniform and equal to $H/80$. The X - Y plane grid was the same as in the RANS simulations. At the inlet, steady state RANS profiles were imposed in the same manner as for the RANS simulations.

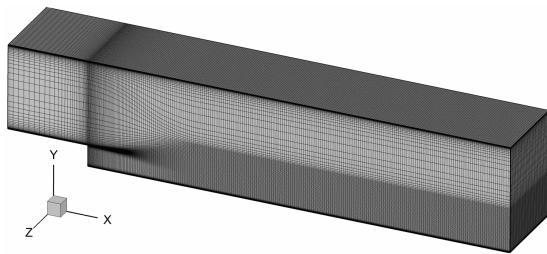


Figure 10. Computational grid used for calculations of the backward-facing step flow and heat transfer with the IDDES approach.

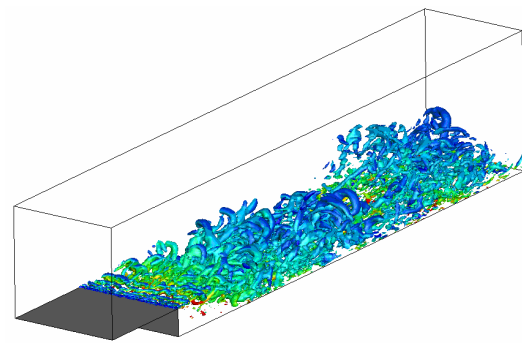


Figure 11. Illustration of resolved vortex structures developing past the backward-facing step: IDDES solution.

5.2. Results of IDDES simulation

The computation was performed with time step equalled to $0.004H/U_{ref}$ so that the Courant number did not exceed a value of 0.2. A sample of about 10,000 time steps was calculated for getting statistics after a transient period.

Instantaneous picture of resolved vortex structures developing past the backward-facing step is visualized in Figure 11 by an iso-surface of the Q -criterion (coloured by temperature). Comparison of mean velocity profiles obtained in our calculations and the experiments [2] is shown in Figure 12. Good agreement between the experimental and numerical results is observed here, at that the two codes used give practically indistinguishable velocity profiles (it should be noted that the RANS simulations show also a good agreement here).

Analysing the computational results for the mean skin friction coefficient and Stanton number distributions (Figures 13 and 14), one can see that, comparing with the experimental data, the IDDES produces a considerably better agreement for the wall friction than in the RANS simulation case, and the two codes used have good coincidence everywhere except a small part of the recirculation zone. As for the Stanton number, the in-house code shows some superiority over Fluent for reproducing the experimental data for $x/H > 4$, but the both codes show certain underestimating the heat transfer rate near the step ($x/H < 4$). Slightly worse results for the Stanton number predicted by Fluent are likely caused by stronger grid sensitivity in heat transfer modelling. Indeed, as shown, for instance, in [6], grid dependency for the Stanton number is stronger expressed than that for the skin friction coefficient.

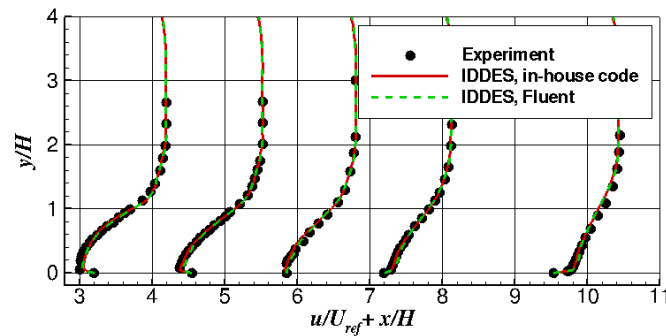


Figure 12. Comparison of mean velocity profiles measured in [2] and computed with the IDDES method. Profiles are plotted at $x/H=3.2, 4.55, 5.86, 7.2, 9.53$

To summarize, the RANS approach gives a better agreement with measurements for the heat transfer while the IDDES technique looks more attractive to predict skin friction accurately. Additional efforts should be made to remove systematic deviation in IDDES predictions of the Stanton number in the part of the recirculation zone adjacent to the step, and capture all the important mechanism responsible for heat transfer there.

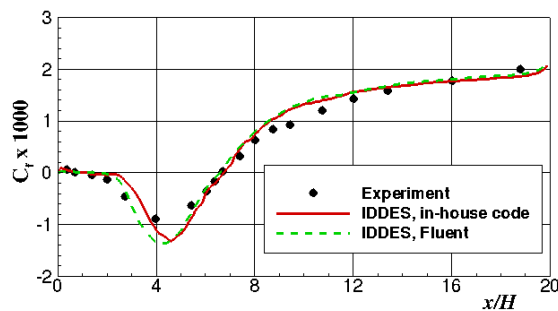


Figure 13. Comparison of the skin friction coefficient distribution measured in [2] and computed with the IDDES method.

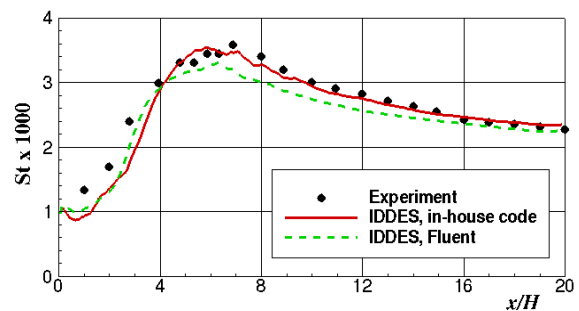


Figure 14. Comparison of the Stanton number distribution measured in [1] and computed with the IDDES method.

6. Conclusions

The 3D RANS-based numerical analysis has been undertaken to evaluate effects of boundary layers developing on the sidewalls of the test section used in recognized experiments devoted to studies of turbulent flow and heat transfer past a backward-facing step. It has been established that the secondary flows arising in the separation region lead to decreasing heat transfer rate on the test section central line by about 4% that is comparable with the measurement errors.

IDDES simulations of flow dynamics and heat transfer have been carried out prescribing periodic conditions in the spanwise direction. Comparing with the experimental data, the IDDES produces a considerably better agreement for the wall friction than the RANS approach, and the two CFD codes used have shown a good coincidence. The IDDES produces the best agreement for the wall friction, whereas the RANS solutions, 2D and 3D, show superiority in predictions of the local Stanton number distribution.

Acknowledgments

The study was supported by the Russian Foundation for Basic Research (grants no. 14-08-00936 and no. 14-07-00065)

References

- [1] Vogel J C and Eaton J K 1985 *J. Heat Transfer* **107** 922
- [2] Adams E W, Johnston J P and Eaton J K 1984 Experiments on the structure of turbulent reattaching flow *Technical Report MD-43 Thermosciences Division, Department of Mechanical Engineering, Stanford University* (Stanford, CA)
- [3] Abrous A and Emery A F 1996 Benchmark computational results for turbulent backward facing step flow with heat transfer *Proc. ASME 31 National Heat Transfer Conf.* vol 9 (ASME HTD-vol 331)
- [4] Abe K, Kondoh T, Nagano Y 1995 *Int. J. Heat Mass Transfer* **38** 1467
- [5] Vieser W, Esch T and Menter F 2002. Heat transfer predictions using advanced two-equation turbulence models *CFX Technical Memorandum CFX-VAL 10/0602* (CFX) pp 26-30
- [6] Rochhausen S, Krüppel F and Fiedler J 2014 *Proc. 11th World Congress. Comp. Mech. (Barcelona, Spain, July 20-25, 2014)* (Barcelona: CIMNE) Vol. 5 p.4939
- [7] Hanjalic K, Laurence D R, Popovac M and Uribe J C 2005 *Proc. ERCOFTAC Int. Symp. on Eng. Turbulence Modelling and Experiments 6*, ed Rodi W, Mulas M (Elsevier Ltd) p 67
- [8] Akselvoll K and Moin P 1995 Large eddy simulation of turbulent confined coannular jets and turbulent flow over a backward facing step *Report TF-63 Department of Mechanical Engineering, Stanford University* (Stanford, CA)
- [9] Keating A, Piomelli U, Bremhorst K and Nesic S 2004 *J. Turbul.* **5** 020
- [10] Shur M L, Spalart P R, Strelets M K and Travin A K 2008 *Int. J. Heat Fluid Flow* **29** 1638.
- [11] Menter F R, Langtry R, Kuntz M 2003 *Turbulence, Heat and Mass Transfer 4* (Begell House) p 625
- [12] Gritskevich M S, Garbaruk A V, Schütze J and Menter F 2012 *Flow, Turbul. Combust.* **88** 431