ASSESSMENT OF THE CAPABILITY OF AVAILABLE METHODS OF FLOW FIELD NUMERICAL MODELLING ON THE EXAMPLE OF AIR FLOW THROUGH A CHANNEL WITH A CIRCULAR BUMP

SŁAWOMIR DYKAS, MIROSŁAW MAJKUT, KRYSTIAN SMOŁKA AND MICHAŁ STROZIK

Silesian University of Technology
Institute of Power Engineering and Turbomachinery
Konarskiego 18, 44-100 Gliwice, Poland

(received: 19 May 2017; revised: 19 June 2017; accepted: 23 June 2017; published online: 7 July 2017)

Abstract: This paper is an attempt at a systematic approach to the flow field analysis in the case of air flowing through a straight channel with a circular bump. The results of the authors' own experimental measurements are compared to those obtained by means of both an in-house and a commercial CFD code. Apart from the RANS method, which is commonly used in engineering applications, a decision was made to use the URANS and LES methods, which are available both in the in-house academic code and in the commercial program. The comparison between the calculation results is performed using what is referred to as the numerical Schlieren image, which for the RANS calculations is compared to the Schlieren image obtained from experimental testing.

Keywords: circular bump, CFD, experiment, air flow

DOI: https://doi.org/10.17466/tq2017/21.3/r

1. Introduction

In the analysis of compressible fluid flows, practically each flow is turbulent even though a laminar flow or an area of the laminar-to-turbulent transition may locally occur on surfaces of solid bodies. The growing demand for effective modelling of such flows resulted in sudden development of first academic and then commercial CFD codes. Due to the turbulent nature of most gas flows characterized by high velocities, the development of CFD methods focused mainly on modelling non-stationary effects caused primarily by the flow turbulence. Conventional Reynolds-averaged Navier-Stokes (RANS) equations based on different viscous turbulence models are intended for modelling stationary flows, and most viscous turbulence models have been developed to analyse problems related to continuous flows. Alternative methods of the non-stationary flow analysis in industrial applications are most often based on the URANS method, *i.e.* on the RANS equations integrated with respect to time at a constant time step which is identical for the entire computational domain. Considering the advancements in computer technology, the large-eddy simulation (LES) methods appear as an effective and efficient alternative tool for modelling turbulent flows. In recent years, the LES techniques have been applied to an ever-increasing number of engineering problems. It should be admitted though that in their overwhelming majority the applications concerned flows of incompressible fluids, *e.g.* gases flowing with low velocities [1, 2]. The LES method has emerged as a perfect tool for modelling noise generated by turbulent flows, *i.e.* for computational aeroacoustics (CAA) issues [3]. Only small eddies are modelled in the LES method, whereas the large ones are solved directly from the flow conservation equations, which means that for geometries and for Reynolds numbers which are significant for engineering applications, the numerical mesh must be made of a very large number of nodes (control volumes).

The aim of this paper is to assess the in-house academic CFD code developed in the Institute of Power Engineering and Turbomachinery (IPET) of the Silesian University of Technology (SUT) for many years [4, 5] and the most popular commercial code used in engineering applications – the Ansys CFX package – on the example of the fluid flow through a channel with a circular bump. Three currently most popular CFD techniques of the flow field identification in the air transonic flow, *i.e.* the RANS, the URANS and the LES methods, will be checked in terms of their usefulness. Additionally, for the geometry under consideration, results are known of experimental testing performed in the laboratory of the SUT Institute of Power Engineering and Turbomachinery – the static pressure distributions and the flow field image obtained using the Schlieren technique.

2. Geometry under analysis

A geometry which is very well suited for the assessment of the flow field where the supersonic-to-subsonic flow transition occurs on the normal shock wave, the position of which is strongly affected by its interaction with the boundary layer, is the geometry of a straight channel with a circular bump. This particular geometry (*cf.* Figure 1) combines the properties of the structure of both the internal flow in nozzles (the shock wave interaction with the walls limiting the flow) and the external flow (around an aerofoil profile). The basic characteristic dimensions (in mm) of the analysed geometry of a channel with a circular bump are presented in Figure 1. The two-dimensional geometry thickness is 10 mm.

Figure 2 presents the computational domain together with an example numerical mesh which is a structured mesh made of three blocks. The mesh is prepared in the ICEM CFD program as a 3D one. The number of control volumes

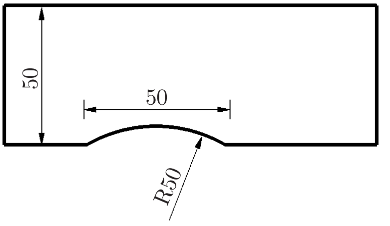


Figure 1. Analysed geometry of the flow channel with main dimensions (in mm)

in the channel depth direction is 3 for modelling by means of the RANS or the URANS technique and 15 for the LES modelling. The calculations are performed assuming the symmetry boundary condition on the channel lateral walls, and the adiabatic wall boundary condition on the bottom and the top wall.

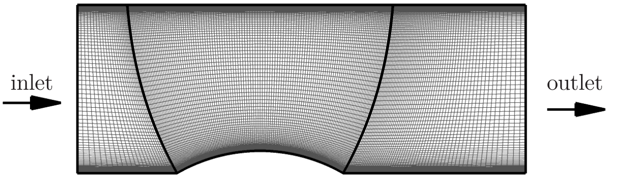


Figure 2. Computational domain under analysis and example computational mesh in plane XY

The mesh size in plane XY was different depending on the selected numerical method: ~ 35 k and 100 k nodes for the RANS and the URANS technique, respectively. The number of the mesh nodes (N) required to capture all length and time scales, which ranges from those comparable to the geometry characteristic dimension l (in this case, it can be the circular bump length of 50 mm, which may at the same time be equivalent to the size of the largest eddy) to the Kolmogorov scales of η, is very high. For one direction, it can be approximately defined as:

$$N = \frac{l}{\eta} \tag{1}$$

where the Kolmogorov length scale depends on kinematic viscosity ν, and the rate of dissipation of turbulence kinetic energy ε.

$$\eta = \left(\frac{\nu^3}{\varepsilon}\right)^{1/4} \tag{2}$$

The turbulence kinetic energy dissipation rate is expressed by the following formula:

$$\varepsilon = \frac{{u'}^3}{l} \tag{3}$$

where u' is the velocity fluctuation determined from the definition of turbulence kinetic energy k, assuming turbulence isotropy:

$$u' = \left(\frac{2k}{3}\right)^{1/2} \tag{4}$$

Ultimately, the number of the mesh nodes for a single direction can be expressed as:

$$N = \left(\frac{u' l}{\nu}\right)^{3/4} = (\mathrm{Re}_t)^{3/4} \tag{5}$$

which shows that it is determined based on the turbulent Reynolds number Re_t.

For $l = 0.05$ m, $\nu = 1.51 \cdot 10^{-5}$ m^2/s and $u' \sim 50$ m/s calculated from the turbulence kinetic energy from the RANS solution, this gives the approximate value of the turbulent Reynolds number $\mathrm{Re}_t = 1.6 \cdot 10^5$, which for the two directions on plane XY gives $N^2 \sim 67\,000$ k nodes, and – in the case of 3D modelling – more than 500 billion nodes. Obviously, this is the number needed for modelling by means of the Direct Numerical Simulation (DNS) method. The following relation is known between the number of nodes necessary for the LES method and defined based on the number of nodes for the DNS for 3D modelling [6]:

$$N_{\mathrm{LES}} > \left(\frac{0.41}{\mathrm{Re}_t^{1/4}}\right)^3 D_{\mathrm{DNS}} \tag{6}$$

It indicates that 700 k nodes adopted for the LES modelling for each XY plane are enough and should ensure the correct solution. In the mesh prepared for modelling by means of the LES method, the number of nodes in the direction of the channel width is not big enough (only 15). It should therefore be expected that the three-dimensional nature of the turbulent structures arising in the flow will not be fully reflected.

For the numerical meshes prepared for the URANS and LES calculations, the value of the dimensionless distance from the wall in the boundary layer $y+$ was smaller than 1, whereas for the RANS calculations – this value was a little higher than 1.

For all the CFD calculations and comparisons between the in-house academic code and the Ansys CFX package, identical computational meshes generated in the ICEM CFD program are used. The calculations for the channel with a circular bump are performed at the same boundary conditions at the inlet, pressure and total temperature $p_0 = 98.38$ kPa and $t_0 = 18\,^{\circ}$C, respectively, and at the mean static pressure value at the outlet of 67 kPa. These conditions correspond to those prevailing during the measurements. The calculations do not take account of condensation of the water vapour contained in air, which is acceptable only for small relative humidity values of about 20%.

In order to verify the results, a decision was made to compare the flow field images obtained using the Schlieren technique, where for CFD computations this corresponds to the value $|\nabla \cdot \rho| = \sqrt{\left(\frac{\partial \rho}{\partial x}\right)^2 + \left(\frac{\partial \rho}{\partial y}\right)^2}$, *i.e.* what is referred to as the numerical Schlieren image. All the calculation results are presented in the range of -100 up to 100 kg/m^4 with step 2 kg/m^4.

3. Calculations

3.1. RANS method

Both the applied academic code and the commercial program solve the Reynolds averaged Navier-Stokes equations for the compressible flow using the Favre averaging method [7, 8]. The equations are supplemented with the two-equation SST k-ω model [9], which is a hybrid solution now generally favoured in engineering calculations. It constitutes a smooth transition from the standard k-ω model used in the boundary layer to the k-ε model as the distance from the surfaces limiting the flow increases. The model includes a modified formulation of turbulent viscosity to take account of the effect of the transport of principal shear stresses.

The flow field solution obtained by means of the RANS method is always a stationary one, *i.e.* the method does not model non-stationary effects arising in the flow, for example due to turbulence or interaction of shock waves.

Figure 3 presents the distributions of static pressure measured during the experiment and the pressure distributions obtained from the CFD calculations performed by means of the academic and the commercial code. It can be seen clearly that the pressure distributions demonstrate very good agreement, which means that the RANS modelling reflects very well the distribution of the measured static pressure, which was averaged in the time of 1 s during the measurements.

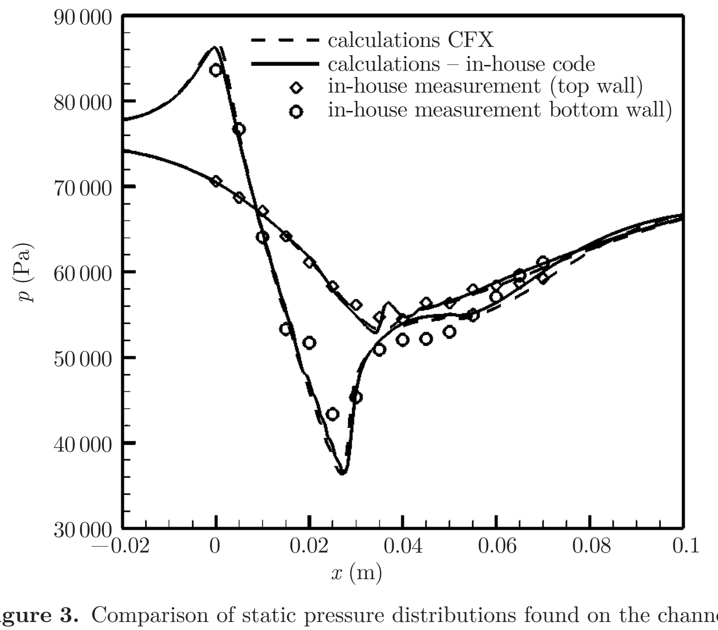


Figure 3. Comparison of static pressure distributions found on the channel top and bottom walls

The good capability of the RANS method combined with the viscous turbulence model for engineering applications is also confirmed by the comparison of the flow field images obtained by means of the Schlieren technique (*cf.* Figure 4).

The comparison shows that the shock wave position obtained from the CFD calculations performed by means of both the commercial and the academic code corresponds to the position measured during experimental testing.

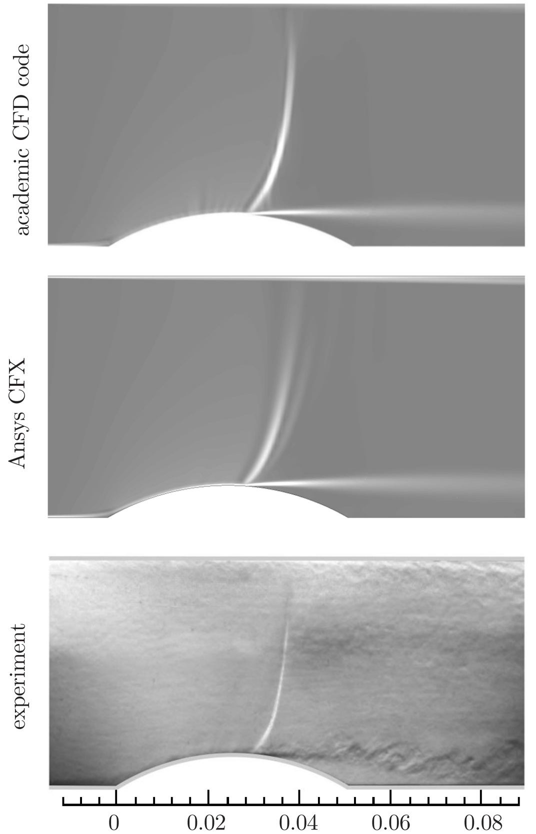
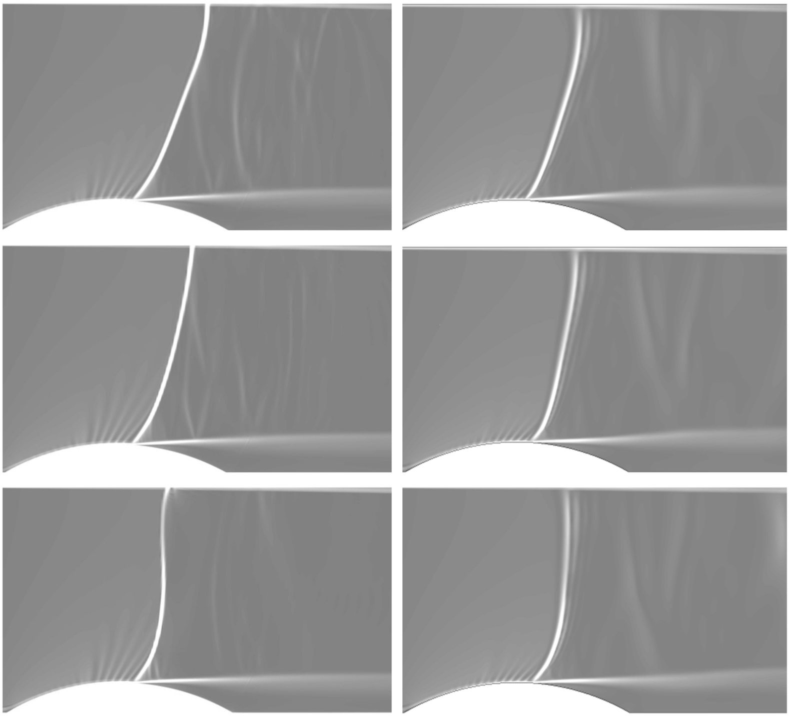


Figure 4. Comparison of the flow field images obtained using the Schlieren technique for the RANS calculations and for the experiment

3.2. URANS method

The URANS method is based on the same computational model as the RANS method, *i.e.* identical boundary conditions, numerical method and turbulence model are applied. The computations are performed with a constant time step of 10^{-9} s for the academic code and 10^{-7} s for the Ansys CFX software. A smaller global time step is used for the commercial code due to the different numerical method of integrating the conservation equations with respect to time.

Figure 5 presents a comparison between the URANS computations using the academic and the commercial code in the form of the flow field momentary images separated from each other in time by 0.1 ms. Owing to the fact that the applied computational mesh is almost three times denser compared to the RANS calculations, the shock wave is much clearer. In non-stationary calculations, the shock wave fluctuations are observed – the wave is fixed to the circular bump in a constant place and its upper part fluctuates from the left to the right with a frequency of about 2.5–3 kHz. This results in generation of compression and expansion waves downstream the shock wave towards the channel outlet, which can be seen clearly in Figure 5.



academic CFD code Ansys CFX

Figure 5. Comparison of the flow field images obtained using the Schlieren technique from the URANS calculations for three moments in time separated by $\Delta t = 10^{-4}$ s

3.3. LES method

The large-eddy simulation (LES) method locates itself in between the URANS and the DNS methods as a compromise between the requirements imposed by the turbulent flow complex structure and the current available computational capabilities. The LES method fundamental assumption is the division of the continuous spectrum of the energy of turbulent fluctuations into a part which is solved (numerically) and a part which is modelled (analytically).

The role of the turbulence model (referred to as the subgrid model) adopted for modelling eddies with smaller length and time scales is very important because such eddies account for up to 20% of the total energy of the turbulent fluctuations arising in the flow. Many modified subgrid models are now in use [2]. Under the LES approach, the form of the Navier-Stokes equations for the compressible flow is obtained through decomposition of the variables present in the equations into the solvable part, mass-averaged using the Favre averaging ($\tilde{}$), very much like in the case of the RANS equations, and the unsolvable part ($'$), which in this case is modelled using the subgrid scale (SGS) model proposed by Smagorinsky [10]. Using the Favre density-weighted filtering [7, 8], the mass, momentum and energy conservation equations can be written as follows:

$$\begin{gathered}
\frac{\partial\bar{\rho}}{\partial t}+\frac{\partial\bar{\rho}\tilde{u}_j}{\partial x_j}=0\\
\frac{\partial\bar{\rho}\tilde{u}_i}{\partial t}+\frac{\partial\left(\bar{\rho}\tilde{u}_i\tilde{u}_j+\bar{p}\delta_{i,j}\right)}{\partial x_j}-\frac{\partial\bar{\tau}_{i,j}}{\partial x_j}=\frac{\partial\tau_{i,j}^{\mathrm{SGS}}}{\partial x_j}\\
\frac{\partial\bar{\rho}\tilde{E}}{\partial t}+\frac{\partial\left(\bar{\rho}\tilde{E}+\bar{p}\right)\tilde{u}_j}{\partial x_j}+\frac{\partial\tilde{q}_j}{\partial x_j}-\frac{\partial\bar{\tau}_{i,j}\tilde{u}_j}{\partial x_j}=\frac{\partial\tau_{i,j}^{\mathrm{SGS}}\tilde{u}_j}{\partial x_j}
\end{gathered}\tag{7}$$

where E is total internal energy. The applied sub-grid model is the Smagorinsky SGS model, for which the following relations are valid:

$$\tau_{ij}^{\mathrm{SGS}}=2\mu_{\mathrm{SGS}}\left(S_{i,j}-\frac{1}{3}S_{k,k}\delta_{i,j}\right)\tag{8}$$

$$\mu_{\mathrm{SGS}}=\bar{\rho}C_s^2V^{2/3}\sqrt{2S_{i,j}S_{i,j}}\tag{9}$$

$$S_{i,j}=\frac{1}{2}\left(\frac{\partial\tilde{u}_i}{\partial x_j}+\frac{\partial\tilde{u}_j}{\partial x_i}\right)\tag{10}$$

$$\tilde{q}_i=-\tilde{\lambda}\frac{\partial\tilde{T}}{\partial x_i}\tag{11}$$

where μ_{SGS} is eddy viscosity, C (in the academic code $C=0.1$) is what is referred to as the Smagorinsky constant, and V is the numerical cell volume.

The same LES method based on the Smagorinsky model is selected for the Ansys CFX commercial code.

Like in the case of the URANS method, the LES computations using the in-house academic code and the commercial program are conducted with the time step of 10^{-9} s and 10^{-7} s, respectively. The comparison between the momentary Schlieren images obtained by means of the two codes is presented in Figure 6. The images are time-separated from each other by 50 μs and no periodicity can be identified in them. In the case of the calculations performed using the academic code, the position of the shock wave arising on the circular bump can be seen clearly, whereas for the Ansys CFX commercial code the wave is deformed considerably. In both cases downstream the circular bump towards the outlet, very complex turbulent structures, eddies with different length and time scales, can be observed.

Additionally, using the commercial code, it was decided to perform the calculations by means of other LES methods included in the package, *i.e.* the LES

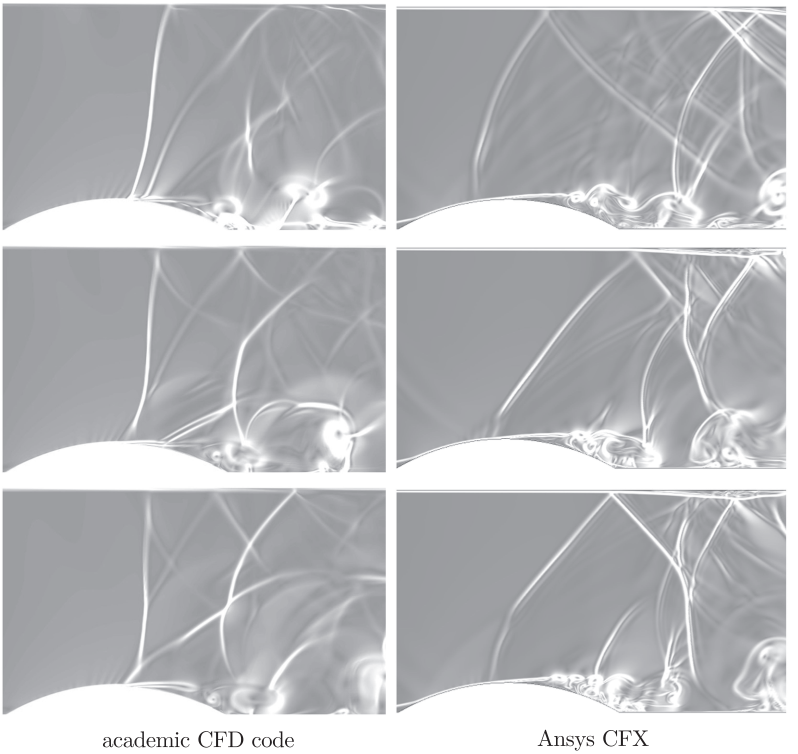


Figure 6. Comparison of the flow field images obtained using the Schlieren technique from the LES calculations for three moments in time separated by $\Delta t = 5 \cdot 10^{-5}$ s

Dynamic Model and the LES WALE Model. But for both these methods a similar tendency was observed for the shock wave to fade on the circular bump crest.

4. Conclusions

The paper presents an analysis of the flow field in a straight channel with a circular bump for a single working point. The analysis is conducted using the authors' own experimental measurements, an in-house academic CFD code and the Ansys CFX commercial package. The in-house and the commercial code computations are performed on the same computational meshes and under identical boundary conditions.

The RANS and the URANS calculation results appear to be very close to the measured averaged pressure values and to the Schlieren image obtained from the measurements. The results obtained by means of the LES method using either the in-house or the commercial code are somewhat disappointing, probably due to a small grid resolution, especially in the Z direction. It seems that the substantially longer computation time is not compensated by the computation quality. The obtained flow field structures, even if averaged over a long period of time, diverge substantially from those given by the measurements or calculations using the RANS or the URANS method. This means that, if we are not interested in the information on fluctuations (eddies) with very small time and length scales, the analysis can be successfully narrowed down by selecting the URANS or even the RANS method only.

Acknowledgements

The presented work was supported from Statutory Research Funds of the Silesian University of Technology.

References

[1] Vreman B, Geurts B and Kuerten H 1997 *Journal of Fluid Mechanics* **339** 357

[2] Froehlich J 2006 *Large Eddy Simulation turbulenter*, Stroemungen Teubner Verlag

[3] Dykas S, Wróblewski W, Rulik S and Chmielniak T 2010 *Archives of Acoustics* **35** (1) 35

[4] Dykas S, Majkut M, Smołka K, Strozik M and Szymański A 2017 *Proceedings of the 12th European Conference on Turbomachinery Fluid Dynamics and Thermodynamics*, Stockholm, Sweden, 3–7 April 2017

[5] Dykas S, Szymański A and Majkut M 2017 *Journal of Thermal Science* **26** (3) 214

[6] Ansys® 2017 *CFX-Solver Theory Guide, ANSYS 17.1*

[7] Favre A 1965 *J. Mec.* **4** 361

[8] Favre A 1965 *J. Mec.* **4** 391

[9] Menter F R 1996 *Journal of Fluids Engineering* **118** 514

[10] Smagorinsky J 1963 *Mon. Weather Rev* **91** 99