Computation of turbulent compressible and incompressible flows

Petr Louda\(^1\) Karel Kozel\(^1\) Jaromír Příhoda\(^2\)

\(^1\) Dept. of Technical Mathematics CTU Prague, Karlovo nám. 13, CZ-121 35 Prague 2
\(^2\) Institute of Thermomechanics AS CR, Dolejšíkova 5, CZ-182 00 Prague 8

The work deals with numerical solution of turbulent steady compressible, steady incompressible, and unsteady incompressible flow. The mathematical model is based on averaged Navier-Stokes equations and artificial compressibility method for incompressible cases. Discretization is implicit, higher order upwind (AUSM). The results are shown for subsonic and transonic flow through a turbine cascade, incompressible flow over backward facing step, and incompressible synthetic jet. Advantages of using explicit algebraic Reynolds stress model (EARSM) over eddy viscosity (SST model) are discussed.

Keywords: synthetic jet, EARSM turbulence model, backward facing step, turbine cascade, artificial compressibility, dual time stepping

Introduction

The work deals with numerical simulation of 2D and 3D turbulent flows governed by averaged Navier-Stokes equations. For both compressible and incompressible flows, a similar system of equations as well as solution procedure is used. In spite of this, in incompressible cases an artificial compressibility method is used. The turbulence models are explicit algebraic Reynolds stress model (EARSM) \([8, 4]\) and eddy viscosity SST model \([6]\). The method of artificial compressibility is further extended for unsteady simulations by means of dual time. All methods considered are implicit, formulated for structured multi-block grids. The spatial discretization uses cell centered finite volume method with higher order upwind interpolation for convective terms.

The computed cases are 2D subsonic and transonic flow through a turbine cascade, 3D flow over backward facing step, and synthetic (unsteady) round jet. The advantage of more complex EARSM model can be seen on the behavior in the shock wave and secondary flow in the channel with step. The SST model is applied to unsteady simulation as well, with acceptable results.

Mathematical model

The model of turbulent flow is based on averaged Navier-Stokes equations. The mean density is obtained by the Reynolds (ensemble) averaging, whereas for velocity and internal energy the Favre (density-weighted) averaging is applied. For a volume \(V\), the system can be expressed by

\[
\int_V \frac{\partial W}{\partial t} + \oint_{\partial V} (F^I - F^V) dS = 0, \tag{1}
\]

where \(F^I\) is inviscid and \(F^V\) viscous flux through the boundary of \(V\). In the model of compressible flow, the system consists of continuity equation, momentum equations, and energy equation:

\[
W = [\rho, u_1, u_2, u_3, \rho E]^T, \tag{2}
\]

\[
F^I = u_c[\rho, \rho u_1, \rho u_2, \rho u_3, \rho H]^T
+ [0, pn_1, pn_2, pn_3, 0]^T,
\]

\[
F^V = \text{col}[0, \tau_{1j}, \tau_{2j}, \tau_{3j}, \tau_{ij} u_i - q_j]^T n_j,
\]

where

\[
E = \frac{1}{\gamma - 1} \rho + \frac{1}{2} u_i u_i + k, \tag{3}
\]

\[
H = E + \frac{p}{\rho}, \quad u_c = u_i n_i
\]
is total energy, total enthalpy and normal velocity, respectively. The turbulent energy is denoted \( k \). The stress tensor \( \tau_{ij} \) is given by

\[
\tau_{ij} = \mu 2 S_{ij} + \tau_{ij}^t, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) = \frac{1}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij},
\]

where \( \mu \) is viscosity and \( \delta_{ij} \) Kronecker delta. The heat flux is given by

\[
q_i = -\frac{\gamma}{\gamma - 1} \frac{\mu}{\rho} \frac{\partial (p/\rho)}{\partial x_i} + q_i^t.
\]

The model for incompressible isothermal flow consists of continuity and momentum equations under assumption that \( \rho = \text{const} \):

\[
W = \begin{bmatrix} 0, u_1, u_2, u_3 \end{bmatrix}^T,
\]

\[
F_I = u_c \begin{bmatrix} 1, u_1, u_2, u_3 \end{bmatrix}^T + \begin{bmatrix} 0, p/\rho, p/\rho, p/\rho \end{bmatrix}^T,
\]

\[
F^V = \text{col}[0, \tau_{1j}, \tau_{2j}, \tau_{3j}]^T n_j.
\]

The terms \( \tau_{ij}^t \) and \( q_i^t \) denote turbulent stress tensor and turbulent heat flux. Two turbulence models are considered: SST eddy viscosity model [6] and explicit algebraic Reynolds stress model due to Wallin [8]. In SST model, eddy viscosity and turbulent Prandtl number \( Pr_t \approx 0.91 \) are used to express turbulent terms by means of the Boussinesq hypothesis. EARSM model of \( \tau_{ij}^t \) contains terms up to fourth order in terms of velocity gradients. The turbulent heat flux again uses \( Pr_t = 0.91 \) and a suitably defined eddy viscosity [8]. Both turbulence models solve system of \( k-\omega \) equations for turbulent scales. The systems are very similar, therefore the differences in results might be attributed to the constitutive relation for Reynolds stress.

**Numerical method**

The averaged Navier-Stokes equations are used in similar form with hyperbolic inviscid part both for compressible and incompressible simulations. In the incompressible case, the artificial compressibility method described below is used. The method allows several kinds of extension to unsteady simulations. We consider here the extension by introducing dual time.

The discretization in space uses cell-centered finite volume method with quadrilateral or hexahedral finite volumes in 2D and 3D case, respectively. The derivatives of velocity or temperature on the boundary of finite volume are computed as mean value in a dual finite volume using vertices of the boundary and centres of adjacent primary finite volumes. The space-discretized system (1) can be written

\[
\frac{\partial W_{i,j,k}}{\partial t} + \text{Rez}(W)_{i,j,k} = 0.
\]

For steady simulations, backward Euler implicit method is used. The three-layer second order accurate implicit method for unsteady simulations is described below.

**Compressible case**

The inviscid flux \( F^I \) is approximated by AUSM splitting [5], where the flux is function of state on both sides of the face, i.e. \( F^I = F^I(W_L, W_R) \). For higher order accuracy, the linear extrapolation for variable \( W \) with limiter was applied. Considering e.g. the face \( i+1/2 \) between cells \( (i,j) \) and \( (i+1,j) \) we have for left and right state \( W_L \) and \( W_R \):

\[
W_L = W_{i,j} + \frac{1}{2} \Psi(r_L) \Delta^-, \\
W_R = W_{i+1,j} - \frac{1}{2} \Psi(r_R) \Delta^+,
\]

\[
\Psi(r) = \frac{r + |r|}{r + 1}, \quad r_L = \frac{\Delta + \epsilon}{\Delta^- + \epsilon}, \quad r_R = \frac{\Delta + \epsilon}{\Delta^+ + \epsilon}, \quad \Delta^- = W_i - W_{i-1}, \quad \Delta = W_{i+1} - W_i,
\]

\[
\Delta^+ = W_{i+2} - W_{i+1}, \quad \epsilon = 10^{-17},
\]

where \( \Psi \) is the van Leer limiter, switching to first order upwind at occurrence of a local extremum of \( W \) (adjacent slopes \( \Delta \) have opposite sign). The interpolation does not depend on grid geometry, which might seem inadequate for turbulent simulations on highly stretched grids. Nevertheless the inviscid flux in a boundary layer is less important than the viscous one, whereas outside of boundary layer sensitivity to sometimes unavoidable imperfections of structured grid is reduced in this way.

**Incompressible case**

In an artificial compressibility method, pressure time derivative is added to the continuity equation:

\[
\frac{1}{\beta^2} \frac{\partial (p/\rho)}{\partial t} + \frac{\partial u_i}{\partial x_i} = 0,
\]

where \( \beta^2 \) is positive parameter. The inviscid part of modified equations is now fully hyperbolic and can be solved by standard methods for hyperbolic conservation laws. In this form, the method is suitable for steady simulations, where \( \partial (p/\rho)/\partial t \rightarrow 0 \) as \( t \rightarrow \infty \). The extension to unsteady flows is given below. The artificial compressibility parameter is chosen so as the flow has mathematically character of subsonic
flow. Therefore, the upwinding is used for convective term only and pressure gradient is computed using central approximation. The convective term can be written

$$F_{c}^{i+1/2} = (u_{i}W)_{i+1/2},$$

$$u_{c}^{i+1/2} = \frac{1}{2}(u_{i} + u_{i+1})n_{1}$$

$$+ \frac{1}{2}(v_{i} + v_{i+1})n_{2} + \frac{1}{2}(w_{i} + w_{i+1})n_{3},$$

where $(n_{1}, n_{2}, n_{3})$ is normal vector of the interface $i + 1/2$. Upwinding is achieved by $\kappa$-interpolation (van Leer), where

$$W_{i+1/2} = \begin{cases} W_{i} + \frac{1}{2}(1 + \kappa)(W_{i+1} - W_{i}) + \frac{1}{2}(1 - \kappa)(W_{i} - W_{i-1}) & \text{for } u_{c,i+1/2} > 0, \\ W_{i+1} - \frac{1}{2}(1 + \kappa)(W_{i+1} - W_{i}) - \frac{1}{2}(1 - \kappa)(W_{i+2} - W_{i+1}) & \text{for } u_{c,i+1/2} < 0. \end{cases}$$

The use of limiters is not necessary. However, to achieve pressure-velocity coupling, pressure diffusion is added to the right hand side of continuity equation [2].

**Unsteady case**

In this work, the simulations of unsteady incompressible flow are considered as well. The artificial compressibility method is used in artificial time $\tau$ and for physical time, second order accurate three-layer implicit scheme is used. The semi-discretized equations can be written

$$\frac{W_{i,j,k}^{n+1} - W_{i,j,k}^{n}}{\Delta \tau} + R \frac{3W_{i,j,k}^{n+1} - 4W_{i,j,k}^{n} + W_{i,j,k}^{n-1}}{2\Delta t} + Re \xi (W_{i,j,k})^{n+1} = 0.$$  

The iterative process denoted by $\nu$ achieves steady state $W_{i,j,k}^{n+1} = W_{i,j,k}^{n}$ at each physical time level.

**Results for steady compressible case**

Authors considered one of the QNET database test-cases, namely the flow in the steam turbine rotor cascade SE1050. Results are shown for subsonic regime with outlet isentropic Mach number $M_{2} = 0.716$ and transonic regime with outlet isentropic Mach number $M_{2} = 1.198$ [9] are shown. Outlet Reynolds number based on chord length was around $Re = 1.5 \times 10^{6}$. The finite volume grid is of H-type (approx. 34,000 finite volumes) and refined around the blade giving aspect ratio up to approx. 800.

The results for subsonic regime are shown in Figs. 1, 2 in the form of isolines of Mach number next to interferometric measurement [9] and surface pressure and friction distribution. The results for transonic flow are shown in the same form in Figs. 3 and 4. Both results compare well with measurements as well as results of several different schemes [3].

The differences by turbulence modeling using SST or EARSM model are negligible for Mach number and surface pressure distribution. Fig. 4. Slightly different is surface friction shown in same figure. EARSM model predicts slightly higher friction on suction side behind the shock wave. More remarkable is difference in turbulent energy as shown in Fig. 5. The SST model most likely over-predicts the production of turbulent energy in the shock wave, due to the non-physical, always positive contribution of normal strain. This is analogous problem as the over-prediction at the leading edge. The EARSM model behaves better in these zones.

The energy loss coefficient $\xi$ is defined as

$$\xi = 100(1 - \gamma^{2/3} \lambda_{\text{iso}}^{2}),$$

$$\lambda_{\text{iso}}^{2} = \frac{\gamma + 1}{2} \left[1 + \frac{\gamma - 1}{2} M_{2}^{2}\right]^{-1} M_{2}^{2},$$

$$\lambda_{\text{iso}}^{2} = \frac{\gamma + 1}{\gamma - 1} \left[1 - \frac{p_{2}}{p_{0}} \right]^{\frac{\gamma - 1}{\gamma - 2}},$$

where $M_{2}$, $p_{2}$ are mean values at line located 1/3 of the pitch downstream of the trailing edge. The computations give $\xi \approx 2.8\%$ and 2.7\% in subsonic and transonic regime respectively, whereas measurements give 3.75\% and 4.7\% respectively [9]. Although it should be remarked this parameter is extremely sensitive to details of evaluation, poor agreement (even opposite Mach number dependence) with measurements is apparent. To some extend this can be attributed to most likely unsteady wake behind the blade, which has not been captured in this simulation.

**Results for steady incompressible case**

In this section, the flow over backward facing step in a channel is presented. The case is taken from Armaly et al. [1], with the width of channel truncated to $6h$, where $h$ is step height. The grid has approx. 290,000 finite volumes, with the minimum finite volume thickness has been $2 \times 10^{-3}h$, $4 \times 10^{-3}h$ and $10^{-2}h$ in vertical, span-wise, and longitudinal direction respectively.

The inlet flow was taken from separate simulation of developed straight channel flow, on the same grid. In a rectangular channel, the secondary flow should be predicted by numerical model. The computed secondary flow is shown in Fig. 6 in the form of velocity vectors. EARSM model at
least qualitatively correctly predicts secondary corner vortices, whereas the eddy viscosity SST model, with isotropic normal stresses, hardly shows any secondary flow, and no secondary vortices. The Fig. 7 shows velocity and turbulent energy in both mid-planes. The velocity in vertical mid-plane is not influenced by the distant corner vortices and is same for both turbulence models. It has been also confirmed that it is same as for 2D developed channel flow. On the other hand, the difference is large in the horizontal mid-plane. The EARSM model gives larger bulk velocity than SST one. Nevertheless, velocity profiles coincide in the middle portion of channel for \(-1 < z/h < 1\), which seems to confirm that channel width is sufficient to make comparisons of vertical mid-plane with 2D simulation.

The velocity in mid-plane is shown for both turbulence models in Fig. 8. The length of separation zone in mid-plane is given in Tab. 1. The inconsistency for SST model is apparent: the SST model agrees well with measurement in 2D case only, the 3D result is unacceptable. The difference between 2D and 3D model is also shown in upper part of Fig. 9. On the other hand, the EARSM model gives consistent results as shown in lower part of same figure.
Results for unsteady incompressible case

In this part, we consider the synthetic jet generated by periodical inflow/outflow with zero mean value in the circular nozzle [7]. The nozzle diameter $D = 8 \text{ mm}$ and Reynolds number $Re = U_{max} D / \nu = 13325$. The computational domain of conical shape is shown in Fig. 10. The length of the domain is $60D$. The problem is solved in Cartesian coordinates. The grid is split to several structured blocks in order to include paraxial zone, see Fig. 10.

The boundary conditions are:

- nozzle ($x = 0$, $\sqrt{y^2 + z^2} \leq D/2$):

\[
\begin{align*}
    u &= U \sin(2\pi ft), \\
    v &= 0.1 v_{Nmn}, \\
    w &= 0.1 w_{Nmn}, \\
    k &= \begin{cases} 
        \frac{3}{2} (uTu)^2 & \text{for } u > 0, \\
        k_{Nmn} & \text{otherwise}
    \end{cases} \\
    \omega &= \begin{cases} 
        k/(5\nu) & \text{for } u > 0, \\
        \omega_{Nmn} & \text{otherwise}
    \end{cases}
\end{align*}
\]
Fig. 5: Isolines of turbulent kinetic energy \( k \). Left: SST model, Right: EARSM model.

\[
\begin{align*}
\text{SST} & \quad \text{scale 14} \\
\text{EARSM} & \quad \text{scale 1}
\end{align*}
\]

where frequency \( f = 75 \text{Hz} \), streamwise amplitude \( U = U_{\text{max}}[1 - (2r/D)^2] \), \( U_{\text{max}} = 27.3 \text{ m/s} \), where \( 0 \leq r \leq D/2 \) is radial position. The other velocity components take into account that entrainment and displacement in experiment induce changes in radial velocity component noticeable already in the nozzle. However the coefficient 0.1 is chosen ad hoc. The \( u_{\text{Nmn}} \), \( w_{\text{Nmn}} \) are values as obtained using homogeneous von Neumann condition. The turbulence level in the nozzle was set to \( T_u = 10\% \).

- wall \((x = 0, \ 0.5D \leq \sqrt{y^2 + z^2} \leq 4.95D)\):
  \[
  u = v = w = 0, \quad k = 0, \quad \omega = \omega_{w},
  \]
  where \( \omega_{w} \) is value of \( \omega \) on the wall. A constant value of \( 2 \cdot 10^3 U_{\text{max}}/D \) was chosen.

- free boundary: This is the rest of domain boundary. Zero Neumann boundary condition was used for velocity, \( k \) and \( \omega \) at \( x = 60D \) and zero derivative in the radial direction for these variables on the conical part.

The turbulence is modeled by SST model. It should be noted that turbulence models are calibrated for steady turbulence only. Typically, the only unsteady phenomenon used for calibration of source terms is decay of grid turbulence, where the mean flow gradients are zero. Further studies are necessary to assess the influence of turbulence model in unsteady simulation. For all simulations we used dual time stepping method with \( \beta = U_{\text{max}} \) and \( \Delta t = 1/72 \) of period, and \( \Delta \tau = 10^7 \Delta t \).

The Fig. 11 shows time averaged velocity on jet axis. The computational result agrees well with experiment [7], except for distance from the nozzle smaller than \( \approx 0.4D \). The computational results achieved using Fluent code with axisymmetrical formulation in [7] are also shown. Inlet boundary condition in Fluent differs from the present one and so does
Fig. 7: Profiles of velocity (above) and turbulent energy (below) in mid-planes of inlet channel

Fig. 8: Velocity profiles in the mid-plane
the velocity near nozzle. The velocity on the axis in Fluent decreases too fast, which suggests higher spreading rate than in experiment. The present computation seems satisfactory here, which means that the turbulence model is acceptable. Next Fig. 12 shows velocity on the axis for several phase angles of inlet excitation. The computed instantaneous velocity corresponds to phase averaged velocity of the experiment [7]. It can be seen that in both cases the unsteadiness reaches up to \( \approx 18 \) nozzle diameters far away. For larger distances, the flow-field corresponds to steady free jet.

**Conclusions**

An implicit finite volume upwind method solving averaged Navier-Stokes equations was presented. It has been applied to steady compressible (sub- and transonic) flows as well as incompressible flows, and extended for unsteady incompressible flows. The turbulence models considered are SST eddy viscosity and explicit algebraic Reynolds stress model. For a subsonic and transonic 2D flow through turbine cascade, numerical results agree very well with measured pressure and friction distribution. The agreement in terms of energy loss coefficient is poor, however high sensitivity of this parameter should be remarked. The advantage of the EARSM model can be seen in more physical turbulence production in shock waves. For 3D steady incompressible flow over backward facing step, the SST model fails to predict secondary flow and shows unacceptable results, although in 2D, the results are good. With EARSM model, the predicted separation length agrees well with measurement. For synthetic jet simulation, the results are in good agreement with measurements despite no corrections were made to SST turbulence model concerning unsteady phenomena.
Acknowledgement

The work was partially supported by grants No. 101/07/1508 and No. 101/05/2681 of GACR and Research Plan No. 6840770010.

References


Fig. 11: Time averaged velocity on the free jet axis

Fig. 12: Phase averaged velocity on jet axis. Left: computation, right: measurement