A Two-Layer Algebraic Turbulence Model for Compressible Flow in Turbomachinery Cascade

A. Dumitrache[1], H. Dumitrescu[1], and F. Frunzulica[1,2]

[1] Institute of Mathematical Statistics and Applied Mathematics, Calea 13 Septembrie no. 13, 050711 Bucharest, Romania,
alexandru.dumitrache@ima.ro, horia.dumitrescu@ima.ro

[2] "POLITEHNICA" University from Bucharest, Bucharest, Romania
ffrunzi@aero.pub.ro

Summary. The objective of this study is to examine the Baldwin-Lomax turbulence model in a finite volume solver to introduce a computer code for complex two-dimensional flows in turbomachinery. The turbulent model was tested with investigating the turbulent flow over a flat plate and other test cases. Three test cases are presented and the computed results are compared with experimental data. The calculated velocity profile agreed well with the experimental data in plate test case and the solver is validated in test case of flow over a semi NACA-0012 airfoil. The solver is used for flow through a multi-blade cascade of an axial compressor in design condition to show its capability of multi-block solution.

1 Introduction

The development of CFD methods has resulted in very useful analysis tools that are able to provide detailed information to enhance the understanding of complex flow physics at design and off-design conditions in compressor/turbine design [3]. The flow calculations have to be carried out on the basis of the averaged Navier-Stokes equations completed with transport equations for turbulence models. One of the groups of statistical turbulence models is the algebraic one or two-layer turbulence closure, but they require the determination of boundary layer parameters to calculate the eddy viscosity. In complex flow such as the flow through a turbine or compressor cascade, the calculation of e.g. shear layer thickness in a CFD code is difficult, because no realistic criterion can be used to define the edge of the boundary layer [2]. That is the specially the case when flow separation exists within the domain.

An algebraic model, which is not written in terms of the boundary layer quantities and is very robust in separated regions, is the standard Baldwin-Lomax (BL) model [1]. The model was modified by Granville [4] and used by

A.D. Fitt et al. (eds.), *Progress in Industrial Mathematics at ECMI 2008*,
Mathematics in Industry 15, DOI 10.1007/978-3-642-12110-4_129,

He [5] for pressure gradient effects. The objective of the present computational study was to examine the BL model in a finite volume solver to introduce a computer code for 2-D flows in turbomachinery studies based on Van Leers flux splitting methods with using of high order limiters.

2 Governing Equations

The integral form of the quasi-three dimensional unsteady Navier-Stokes equations over a moving finite area **A** is

$$\begin{aligned}\frac{\partial}{\partial t}\int\int_{\triangle A} U dxdy + \oint_s [(F - Uu_g - V_x)\mathbf{n}_x + (G - Uv_g - V_y)\mathbf{n}_y] \\ = \int\int_{\triangle A} S dxdy\end{aligned} \tag{1}$$

where

$$\begin{aligned}
U &= h\,[\rho \quad \rho u \quad \rho v r \quad \rho\rho e]^T\,,\\
F &= h\,[\rho \quad \rho uu + p \quad \rho uvr \quad (\rho e + p)u]^T\,,\\
G &= h\,[\rho \quad \rho uv \quad (\rho vv + p)r \quad (\rho e + p)u]^T\,,\\
S &= [0 \quad p\partial h/\partial x \quad 0 \quad 0]^T\,.
\end{aligned}$$

and M^T denotes the transpose of a matrix M.

The quasi-three dimensional effects are introduced by allowing specified variations of r and h in the axial direction. Both u_g and v_g are the moving mesh grid velocities, to rotor blades and blade vibration [6, 7]. In present work, only the former is considered, thus u_g is zero and V_g is equal to the blade rotation velocity. V_x and V_y are the viscous terms:

$$V_x = h\,[0 \quad \tau_{xx} \quad r\tau_{xy} \quad -q_x + u\tau_{xx} + v\tau_{xy}]^T\,, \tag{2}$$

$$V_y = h[0 \quad \tau_{xy} \quad r\tau_{yy} \quad -q_y + u\tau_{xy} + v\tau_{yy}]^T \tag{3}$$

where

$$\tau_{xx} = \frac{2}{3}\mu\left(2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y}\right), \quad \tau_{yy} = \frac{2}{3}\mu\left(2\frac{\partial v}{\partial y} - \frac{\partial u}{\partial x}\right),$$

$$\tau_{xy} = \frac{2}{3}\mu\left(2\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}\right), \quad q_x = -k\frac{\partial T}{\partial x}, \quad q_y = -k\frac{\partial T}{\partial y}.$$

Flow calculations have to be carried out on the basis of the above averaged Navier-Stokes equations in conjunction with transport equations for BL turbulence closure.

2.1 Baldwin-Lomax Turbulence Model

The Baldwin-Lomax turbulence model is a relatively simple algebraic model that makes use of a two-layer diffusivity formulation

$$\mu_t = \begin{cases} \mu_{t_{inner}} & \text{if } y_n \leq y_{crossover} \\ \\ \mu_{t_{outer}} & \text{if } y_n > y_{crossover} \end{cases}$$

where y_n is the normal distance to the wall and $y_{crossover}$ is the minimum value of the y_n at which $\mu_{t_{inner}} = \mu_{t_{outer}}$. In the inner layer, the eddy viscosity coefficient is defined as $\mu_{t_{inner}} = \rho l^2 \Omega$ where $l = k y_n \left[1 - \exp\left(y^+/A^+\right)\right]$ is the length scale of the turbulence in the inner region, k and A^+ are model constants, Ω is the magnitude of the vorticity,

$$\Omega = \left| \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \right|$$

The wall factor is given by $y^+ = \dfrac{\sqrt{\rho_w \tau_w}}{\mu_w} y_n$ where, ρ_w, μ_w and τ_w are the density, molecular viscosity and laminar shear stress at the wall.

In the outer layer, the eddy viscosity is defined by $\mu_{t_{outer}} = K C_{cp} F_{wake} F_{kleb}$ where K and C_{cp} are the model constants, and the function F_{wake} is taken by

$$F_{wake} = \min\left(y_{max} F_{max}, C_{wk} y_{max} U_{max}^2 / F_{max}\right)$$

and

$$F_{kleb} = \left[1 + 5.5 \left(\frac{C_{kleb} y}{y_{max}}\right)^6\right]^{-1}$$

Here, F_{max} is determined by the maximum value of the function $F = y_n \Omega \left[1 - \exp(-y^+/A^+\right]$ and y_{max} is the value of y_n at which this maximum occurs. Also, U_{max} is the maximum difference of the magnitude of the velocity in the profile.

The model constants are given by

$$k = 0.4, \quad A^+ = 26, \quad K = 0.0168$$

$$C_{cp} = 1.6, \quad C_{wk} = 1.0, \quad C_{kleb} = 0.8$$

Transition to turbulence can be modeled by setting a cut off value for the computed eddy diffusivity. The suggested criterion is $\mu_t = 0$, if $\mu_{max} < C_{mutm}$, $C_{mutm} = 14$. For use with multigrid, the turbulence viscosity is evaluated only on the fine mesh and frozen on all coaster meshes.

3 Numerical Schemes and Results

The Van-Leer's method is used for splitting the convective fluxes and central differencing is used for viscous terms. To improve the accuracy of convective terms the Universal Van Leers limiter is used in mid points of computation area and first order accuracy is used for boundary cells. The numerical equations are used with explicit scheme because of its faster convergence history. The stability criteria, is based on characteristic values of Jacobian matrices of convective terms. The non-reflective boundary condition is used for downstream of computational domain to speed up the convergence process, especially in low Mach number flows. A multi-block algorithm is implemented for complicated geometries, to divide the area to multi-channels. Each channel is solved with using neighborhood channel boundary condition in jointed boundaries of channels to give the influence of adjacent zones in being solved area. All channels are solved alternatively form bottom to up and then time increases to new time step of solution. To minimize the convergence history the non-reflective boundary condition [9] is used for downstream of computational domain.

The validation of the implemented turbulence model is first done for the turbulent flow over a flat plate. The calculation are carried out with inviscid inlet boundary conditions located 100% of the plate length upstream of the leading edge with a Mach number of 0.2. The transition point is set to close to the leading edge. The Reynolds number for flow around flat plate is set to 10^6 based on plate length scale. The results give good agreement with experimental data of Wieghardt [10].

In the next test case, the subsonic viscous flow over a semi NACA-0012 airfoil is considered. The Reynolds number is set to 10^6 and the free upstream Mach number is set to 0.4. In the next test case, the transonic viscous flow over NACA-0012 airfoil with a 2° angle of attack is considered and the free stream Mach number is set to 0.82. Figure 1 illustrates the comparison of pressure coefficient distribution for upper and lower surface of airfoil with experimental results. The good agreement of computed results and the experimental data, gives the adequate assurance about the solver, which implements the Van Leers flux splitting scheme and the related high ordering limiter and the BL turbulence model in compressible flows in presence of pressure gradients and shock induced cases.

As the final case study, a two- dimensional cascade of an axial compressor with NACA-65(10) airfoil geometry for rotor and stator blades is considered. Figure 2 shows the pressure contours, of the stage when the rotor is located in face to face situation with respect to stator location. The Mach number is set to 0.2, because many instability phenomenas happens in low Mach numbers for transonic compressors, for example during starting process of a gas turbine engine. Figure 3 shows the pressure contours when the rotor is moved in rotation direction for a half pitch of blades row. The mesh points are clustered for boundary layer capturing. The mesh resolution is such set to have at least a numerical cell in viscous sub-layer of boundary layer in all test cases. This means the mesh resolution is adequate to capture turbulent behavior. The transition point is set at the near of leading edge of the blades in stage problem.

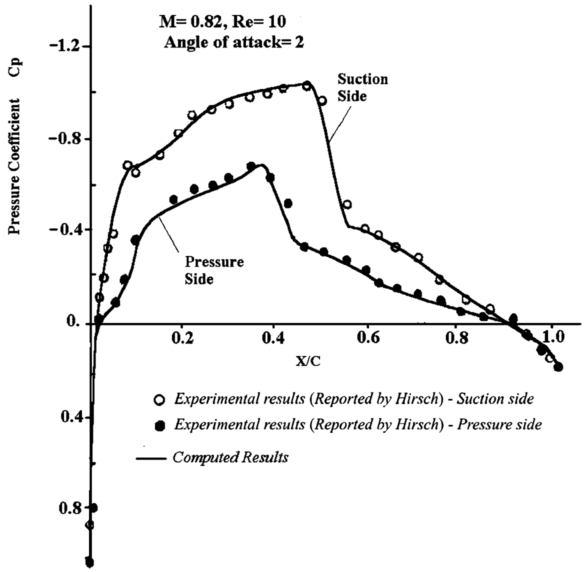

Fig. 1. Computed pressure coefficient over a NACA0012 airfoil at M = 0.82, Re = 10^6 and 2° angle of attack, compared with experimental results [8]

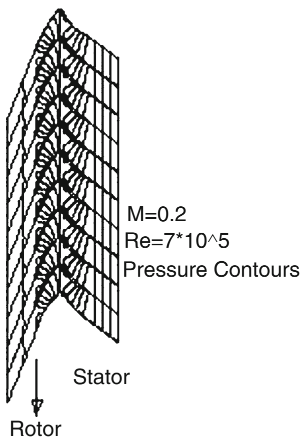

Fig. 2. Pressure contours for a 10-passage-stage of an axial compressor (no. of cells = 320,000)

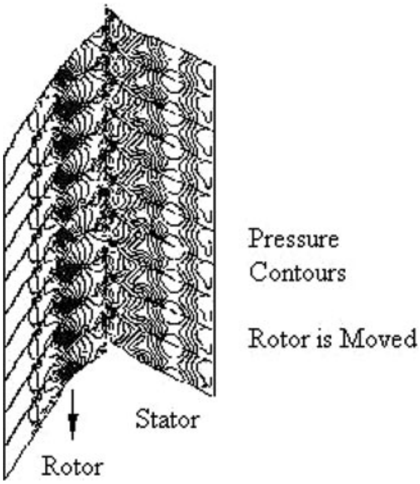

Fig. 3. Pressure contours for a 10-passage-stage of an axial compressor when the rotor is moved about half pitch of cascade

4 Conclusions

A computer code for numerical simulation of the 2-D inviscid and viscous flow in turbomachinery blade channels was presented. The turbulent model was tested with investigating the turbulent flow over a flat plate and other test cases. The calculated velocity profile agreed well with the experimental data in plate test case and the solver is validated in test case of flow over a semi NACA-0012 airfoil. The good agreement of pressure distribution with experimental data, in turbulent flow around NACA-0012 airfoil with angle of attack, gives the robustness of the code and the implemented schemes in transonic flows with the existence of adverse pressure gradients and the interaction of shock and boundary layer. Consequently, a solver with an algebraic turbulent modeling for compressible viscous-inviscid flows in complicated geometries is achieved, which may be useful in 2-D unsteady studies of turbomachinery investigations in off-design conditions.

References

1. Baldwin, B.S., Lomax, H.: AIAA Paper No. 78-257 (1978)
2. Bohn, D., Edmunds, R.: Proceedings of the International Gas Turbine and Aeroengine Congress and Exposition, Houston, 95-GT-90, 1995
3. Deardorff, J.W., J. Fluid Mech. **41**, 453–480 (1970)
4. Granville, P.S.: AIAA J. **25**, 1624–1627 (1987)
5. He, L., J. Comput. Phys. **36**, 55–70 (1980)
6. He, L.: J. Propul. Power **9**, 272–280 (1993)
7. He, L.: Int. J. Comput. Fluid Dyn. **3**, 217–231 (1994)

8. Hirsch, C.: Numerical Computation of Internal and External Flows: Fundamentals of Numerical Discretization, vol. 1. Wiley, New York (1988)
9. Rudy, D.H., Strikwerda, J.C.: J. Propul. Power **13**, 31–38 (1997)
10. Wieghardt, K., Tillmann, W.: Computation of turbulent boundary layers. In: Proceedings of AFOSR-IFP-Stanford Conference, 1968