Aerospace Science and Technology 32 (2014) 84–93

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Verification and validation of Reynolds-averaged Navier–Stokes turbulence models for external flow ☆

Jacob A. Freeman [a,*,1], Christopher J. Roy [b]

[a] United States Air Force, United States
[b] Department of Aerospace & Ocean Engineering, Virginia Tech, 215 Randolph Hall, Blacksburg, VA 24061, USA

ARTICLE INFO

Article history:
Received 26 June 2013
Received in revised form 1 November 2013
Accepted 5 November 2013
Available online 27 November 2013

Keywords:
Verification and validation
Code verification
Solution verification
Reynold-averaged Navier–Stokes turbulence models
Subsonic compressible flow
Computational fluid dynamics

ABSTRACT

The Spalart–Allmaras (S-A) turbulence model in the NASA-Langley CFL3D and FUN3D flow solvers has been previously verified 2nd-order accurate. For low subsonic 2-D applications (turbulent flat plate and NACA 0012 airfoil at $\alpha = 0°$), solutions from the S-A, S-A with Rotation and Curvature (SARC), Menter Shear-Stress Transport (SST), and Wilcox 1998 k-ω turbulence models in commercial flow solvers, Cobalt and RavenCFD, are compared with NASA results for code verification. Of 36 case evaluations, each of which uses 5 systematically refined computational meshes, only 7 approach 2nd-order observed accuracy, but 27 cases show 1st-order or better, indicating the formal order may be less than 2 for these applications. Since Cobalt and RavenCFD turbulence models perform comparable to NASA's verified models and since rigorous code verification is not possible without access to source code, the presented evidence suggests these turbulence models are implemented correctly for these or similar flow conditions and configurations. For solution verification, estimates of numerical uncertainty are less than 0.5% for 94% of the cases and less than 0.1% for 61% of the cases. For validation, the turbulent flat plate solutions match experiment skin friction within 4.8% for $x/L > 0.05$, and for airfoil drag coefficient, S-A and SST agree within 1.2% of experiment, SARC 2%, and k-ω 4%.

Published by Elsevier Masson SAS.

1. Introduction

With initial and subsequent versions of flow solvers, developers and users subject the code to a suite of regression tests and case validations, such as the 1-D Riemann shock-tube problem [14]; in 2-D, an inviscid supersonic inlet with compression ramp [14], inviscid airfoil in transonic flow, inviscid base flow [10], laminar flat plate [14], viscous, turbulent boundary-layer flow for the flat plate, supersonic ramp, and an airfoil [3,4,6,10–12,15,16,21,24,30], or various types of jet and shear flow [1,2]; then well tested 3-D applications such as a supersonic missile with fins [14], a wing [11,30], or a civil air transport in transonic cruise [29]. Often, however, little attention is given to code verification (to determine the code's observed order of accuracy, as compared with its formal order) or to solution verification (to quantify numerical accuracy of the code's predicted solutions) [18]. The preference is to verify a code's formal order of accuracy by computing the solution to a problem with an exact solution. The exact analytical solution or an exact manufactured solution may then be used to accomplish this code verification [18]. Without access to source code, in the case of commercial flow solvers, the user may conduct code verification by carefully comparing results with those from a flow solver that has been rigorously verified. To conduct solution verification, users may then use the verified code to compute solutions for various applications and estimate the numerical error in those solutions, effectively placing "error bars" on the computational predictions. As the last step in the verification and validation process, the user validates the model to assess how accurately the model represents the physical flow; this is accomplished by comparing the computed solution with experimentally obtained data.

The purpose of this study is to verify turbulence models in the commercial flow solvers, Cobalt and RavenCFD, which are derived separately from the Air Force Research Lab's *Cobalt*$_{60}$, by comparing their solutions and behavior with those obtained from the previously verified NASA-Langley flow solvers, CFL3D (cell-centered structured) and FUN3D (node-centered unstructured) [20,22]. The code and solution verification are performed using two subsonic, 2-D turbulent applications: flat plate and NACA 0012 airfoil at angle of attack, $\alpha = 0°$. These applications were selected because their combined features may represent a more complex, 3-D subsonic flow field. These verification activities compare results from four Reynolds-averaged Navier–Stokes (RANS) turbulence models: Spalart–Allmaras (S-A), S-A with corrections for

☆ The views expressed in this article are those of the authors and do not necessarily reflect the official policy or position of the U.S. Air Force, the U.S. Department of Defense, or the U.S. Government.

* Corresponding author. Tel.: +1 3106534117.
E-mail addresses: jacob.freeman@us.af.mil (J.A. Freeman), cjroy@vt.edu (C.J. Roy).
1 Lieutenant Colonel.

1270-9638/$ – see front matter Published by Elsevier Masson SAS.
http://dx.doi.org/10.1016/j.ast.2013.11.004

rotation and curvature (SARC), Wilcox 1998 k-ω, and Menter shear-stress transport (SST). Model validation is not stressed in this study, but there is some validation for each case. While the flow solvers, turbulence models, and test cases are specific to this study, the *process* by which code and solution verification are conducted for commercial code may be broadly applied throughout the computational community.

2. Background and methods

2.1. Flow solvers

Cobalt [7] solves the 3-D unsteady, compressible Euler and Navier–Stokes equations at cell centers, uses the method of finite volumes, and is parallelized. It is designed to use structured or unstructured mesh topologies, including prisms, tetrahedra, and hexahedra in 3-D, or quadrilaterals and triangles in 2-D, all with arbitrary cell skewness, curvature and/or stretching rates. Cobalt combines the exact Riemann solver of Gottlieb and Groth [13] and the approximate Riemann solver of Harten–Lax–van Leer-Contact (HLLC) [26] with a least-squares method to attain 2nd-order spatial accuracy, and it uses a point-implicit method with Newton sub-iterations for 2nd-order temporal accuracy. Its implicit method allows for Courant–Friedrichs–Lewy (CFL) numbers as large as 1×10^6. Cobalt offers eight turbulence models, including the four used for this study, which have formally 2nd-order accurate numerical implementations but may revert to 1st-order in the presence of discontinuities, contact surfaces, large flow gradients, or singularities. Cobalt offers one discontinuous flux limiter which may influence the solution observed order of accuracy [7,14,29].

RavenCFD [8] also solves the 3-D unsteady, compressible Euler and Navier–Stokes equations at cell centers, uses finite volumes, is parallelized, may use structured or unstructured mesh topologies, is formally 2nd-order accurate in space and time, uses Newton sub-iterations with its implicit solver, and allows the following options to users: fully implicit or explicit using a 4-stage Runge–Kutta solver; flux-splitting schemes of either Gottlieb and Groth [13] or Edwards Low-Diffusion (LDFSS) [27], which is designed primarily for reacting and multi-phase flows; local or global time-stepping; wall functions; and various flux limiters [5]. All RavenCFD simulations in this study use the minmod limiter. RavenCFD offers nine turbulence models, including three used for this study, all with 2nd-order numerical implementations that may reduce to 1st-order when exposed to the effects noted above. RavenCFD does not include the SARC turbulence model.

2.2. Turbulence models

To reduce computational mesh cell count and overall computation time for applications with large Reynolds number flow, all scales of turbulent flow are modeled with RANS turbulence models in this study. The turbulence models include the S-A one-equation (meaning one transport partial differential equation), SARC one-equation, Wilcox 1998 k-ω two-equation, and Menter SST two-equation. For the detailed equations and coefficients used by these turbulence models, see [10,20,25]. Care is taken to ensure turbulence model equations and coefficients are common among all flow solvers in this study, including one revision to a RavenCFD SST coefficient for conformity.

The S-A turbulence model is often applied to aircraft applications, including predicting separation due to adverse pressure gradients. The S-A model is a function of velocity, kinematic viscosity, vorticity, and wall distance. In the laminar sub-layer it uses a wall-destruction function to reduce turbulent viscosity, and to transition the boundary layer from laminar to turbulent it includes trip functions. The S-A model relies on 11 empirical constants [10]. The SARC model includes a modification in the production term which is a function of kinematic strain and rotation rates, as well as three additional constants [25]. The S-A and SARC models show good agreement with experiment for subsonic flow over a flat plate, sub- and transonic flow over airfoils and wings, rotating and curved channels, and turbulent shear flow [3,4,6,11,12,15,21,24,30].

Wilcox's 1998 k-ω turbulence model is often used for wall-bounded flow, regions of large separation, and terms were added to better model planar shear layers. The two transport variables are turbulent kinetic energy, k, and turbulent specific dissipation rate, ω, and the model is a function of velocity, kinematic viscosity, and turbulent shear stress. The model includes low Reynolds number corrections for transition from laminar to turbulent boundary layer. The k-ω model also relies on 11 empirical constants [10]. The k-ω model shows good agreement with experiment, though generally not as good as the S-A, SARC, and SST models, for subsonic flow over a flat plate and an airfoil, and turbulent shear flow [2,4,6].

Menter's SST turbulence model combines the accuracy of the k-ω model for wall-bounded flow with that of the k-ε model for shear flow; the 2nd transport variable is turbulence dissipation, ε. Away from the wall, the ε-equation is transformed into an ω-equation, and the model relies on a computationally expensive switching function between the two sub-models. The SST model is a function of velocity, kinematic viscosity, turbulent shear stress, vorticity, and distance from the wall, and it relies on 10 empirical constants [10]. The SST model shows good agreement with experiment for subsonic flow over a flat plate and sub- and transonic flow over airfoils [3,4,6,11,21]. For the comparisons in this study, Cobalt and RavenCFD use the version of SST discussed above, while FUN3D and CFL3D both use (for the flat plate case, not for the airfoil) a variant form of the SST model, noted SST-V. To improve numerical stability, the SST-V model transport equations include a modification to the vorticity source term [20].

2.3. Case descriptions

The unit tests include a 2-D flat plate to verify and validate boundary layer modeling and a 2-D airfoil for adverse or non-zero pressure gradient modeling.

Fig. 1 shows the 2-D flat plate formulation from NASA-Langley [20], which includes five levels of systematically refined structured meshes; Fig. 1(b) shows the 2nd coarsest of those grids, the finest being 545×385. Values for the average y^+ for the first cell along the surface range between 0.68 for the coarsest grid to 0.04 for the finest; y^+ is defined as

$$y^+ = \frac{y}{\nu}\sqrt{\frac{\tau_w}{\rho}} = yRe_{/x}\sqrt{\frac{C_f}{2}} \tag{1}$$

where y for this case is the height of the first cell (m), ν is kinematic viscosity (m^2/s), τ_w is wall shear stress (N/m^2), ρ is density (kg/m^3), $Re_{/x}$ is Reynolds number per length (m^{-1}), and C_f is skin friction coefficient. Freestream Mach, $M_\infty = 0.2$, is selected to ensure essentially incompressible flow, though the flow solvers all use compressible equations. Reynolds number is $Re_{/x} = 5 \times 10^6$ m^{-1}. Specified at the inflow boundary are total pressure, $P_0 = 117{,}684.90$ Pa, and total temperature, $T_0 = 302.4$ K; they are based on specified $T_{ref} = 300$ K and calculated $P_{ref} = 114{,}448.3$ Pa (from given M_∞ and $Re_{/x}$). At the outflow boundary, $P = P_{ref}$. The plate has no thickness, length is $L = 2.0$ m, and skin friction coefficient values are extracted at $x = 0.97008$ m, or $x/L = 0.48504$. A point singularity at the plate leading edge poses a potential problem with the setup; the singularity makes it difficult, particularly for node-centered codes, for observed order of accuracy to