CFD SIMULATION OF LOW SPEED TURBULENT FLOW PROBLEMS USING UNSTEADY RANS AND LARGE EDDY SIMULATION APPROACH

Sekhar Majumdar*, B.N. Rajani**, D.S. Kulkarni** and M.B. Subrahmanya**

Abstract

The present paper focuses on the recent development of an implicit pressure-based finite volume algorithm for numerical solution of Navier Stokes equation in an inertial frame of reference for prediction of unsteady incompressible flow problems. The algorithm uses boundary-conforming, multiblock structured grid with moving boundaries, collocated variable arrangement with momentum equations resolved along cartesian directions, second order accurate spatial and temporal discretisation schemes for the convective fluxes and a pressure-velocity solution strategy. Effect of turbulence is simulated using one of the two different approaches. In the Unsteady Reynolds Averaged (URANS) approach coupled to appropriate eddy viscosity based turbulence models, the Navier Stokes (NS) equations, averaged over the whole range of turbulent length scales of the flow, are solved numerically. On the other hand, in the Large Eddy Simulation (LES) approach, the model filtered 3D NS equations are directly solved for the flow variables to resolve the large scale turbulent motions whereas the transport processes at the fine subgrid scale level only are simulated using simple algebraic turbulence model. The capabilities and limitations of both the cost-effective URANS approach and the relatively expensive but rich in physics LES approach have been demonstrated for few application problems of engineering interest.

Key Words: Multiblock Boundary-Conforming Grid, Moving Boundary, Pressure-Velocity solution strategy, Eddy viscosity based Turbulence Models, Unsteady Flow, Large Eddy Simulation, Smagorinskys SGS model

Introduction

In the last three decades, Computational Fluid Dynamics (CFD) has evolved from a mere academic curiosity to an emerging technology, used extensively today, for computer-aided design and analysis of a wide variety of systems and equipments in industry involving turbulent fluid flow coupled to even other complex transport processes like heat transfer, multiphase flow or chemical reaction etc.. Specially numerical simulation of unsteady flows with one or more moving boundaries are of great interest for the designers to understand the dynamics of various complex flow situations like the behaviour of an aircraft or a naval vessel during maneuvers, flow past helicopter rotors or flow through wind turbine or gas turbine blade passages etc.. In these applications, flows are highly nonlinear due to unsteadiness, flow separation, viscous/inviscid, vortex/body or vortex/vortex kind of interactions, transition to turbulence or relaminarisation. However, considering the constraint of computing resources, URANS methodology, is often used in practice as the most cost-effective approach to predict the mean flow characteristics of the complex turbulent flow systems. Two major arguments against the use of URANS procedure are the loss of many important details of turbulence interaction due to Reynolds averaging and also that almost all the eddy viscosity based turbulence models have been designed and calibrated on the basis of mean flow parameters of turbulent shear flows only. On the other hand, in the moderately expensive LES approach, the complex physics of turbulent flow is resolved more accurately since the motion of the large scale flow structures involved in momentum and energy transfer processes are resolved numerically from

* Former Head, Computational and Theoretical Fluid Dynamics Division, National Aerospace Laboratories, Post Box No. 1779, Bangalore-560 017, India. Presently Professor, Department of Mechanical Engineering, Nitte Meenakshi Institute of Technology, Bangalore-560 064; Consultant CFD, Fluidyn Software and Consultancies Pvt Ltd, Email : essem17@gmail.com
** Scientist, Computational and Theoretical Fluid Dynamics Division, National Aerospace Laboratories, Post Box No. 1779, Bangalore-560 017, India
the 3D unsteady NS equations and the effect of the smallest fine scales of turbulence only are modeled. The present paper provides a very brief overview of the work carried out at the CTFD Division, NAL Bangalore, by the research group of the first author during the last fifteen years on the development of a robust and accurate general-geometry finite volume algorithm for CFD analyses of unsteady low speed flows using either URANS or LES approach for turbulence simulation. The capabilities and limitations of the algorithm are demonstrated through five different interesting application problems.

**Numerical Method and Turbulence Simulation**

**Governing Equations of the URANS Approach**

In an inertial frame of reference, the Reynolds Averaged Navier Stokes equations and the continuity equation for unsteady incompressible flow with moving boundaries may be written in tensor form as following using general non-orthogonal curvilinear coordinates where \( j, k \) and \( m \) as the summing indices; \( \mu \) and \( \rho \) are the fluid viscosity and density respectively; \( p \) and \( \bar{U}_i \) are the time-averaged pressure and cartesian velocity component respectively; \( u_i \) is the corresponding fluctuating velocity component due to turbulence and \( v_i \) is the grid velocity component indicating the motion of the body around which the flow is analysed. \( J \) is the transformation Jacobian between the cartesian and the curvilinear coordinates and \( \beta_i^j \) and \( B_i^j \) are the relevant geometric coefficients related to the transformation.

**Momentum Conservation**:

\[
\frac{\partial (\rho \bar{U}_i)}{\partial t} + \frac{1}{J} \frac{\partial}{\partial x_j} \left[ (\rho \bar{U}_i \bar{U}_j - \bar{v}_i \bar{v}_j) \beta_i^j \right] - \frac{B_{ik}}{J} \frac{\partial x_m}{\partial x_i} B_i^m = 0
\]

(1)

**Mass Conservation**:

\[
\frac{\partial}{\partial x_j} \left( \rho (\bar{U}_i - \bar{v}_i) \beta_i^j \right) = 0
\]

(2)

The Reynolds stress tensor \(- \rho u_i u_k \) is evaluated through appropriate turbulence models. The Linear Eddy Viscosity (LEV) based models, most widely used in URANS computation of complex flows, assume the Reynolds stress tensor components to be directly proportional to the mean strain rates as follows:

\[
- \rho u_i u_k = \frac{\mu_t}{J} \left( \frac{\partial \bar{U}_i}{\partial x_j} \beta_i^j + \frac{\partial \bar{U}_j}{\partial x_i} \beta_i^j \right) - \frac{1}{3} \rho \delta_{ik} u_m u_m
\]

(3)

where, \( \delta_{ij} \) is the Kronecker Delta and the subscript \( m \) is a summation index. The eddy viscosity \( \mu_t \) is evaluated from the relationship with the local turbulence scalars as following:

\[
\mu_t = \rho C_{\mu} E_s T_s
\]

(4)

where \( E_s \) and \( T_s \) are appropriate energy scale and time scale respectively defining the local turbulence level, and \( C_{\mu} \) is a model constant. Five different eddy viscosity based turbulence models are incorporated in the present URANS algorithm. The widely used \( k-\epsilon \) model [10], \( k-\omega \) model [29] and Shear Stress Transport model of Menter [18] assume \( E_s = k \) and \( T_s = k/\epsilon \) or \( 1/\omega \) where \( k \) is the turbulence energy, \( \epsilon \) the dissipation and \( \omega \) the dissipation of turbulence energy. The Spalart-Allmaras model [27], on the other hand, solves a single transport equation for \( \mu_t \) itself instead of using different scalars to evaluate \( \mu_t \). Another advanced LEV based model called V2F, proposed recently by Durbin [5], assumes \( E_s = v^2 \) and \( T_s = \max \left[ \frac{k}{\epsilon}, 6 \left( \frac{\mu}{\omega} \right)^{1/2} \right] \) where \( v^2 \) is a scalar representing the wall-normal component of the turbulence energy near the wall. The modeled transport equations for the relevant turbulence scalars, the closure coefficients, the special damping functions and additional terms for simulation of the near wall effects in \( k-\epsilon \) model are described elsewhere [4] in details.

**Governing Equations for the LES Approach**

In LES approach, appropriate filtering operation [22] is first defined to decompose each flow variable \( \Phi_l \) in the instantaneous NS equation into the sum of a filtered or resolved component \( \Phi_l^f \) and a residual or subgrid scale component \( \Phi_l^s \). The present algorithm uses a box filter as the filter kernel. The model filtered equations are solved numerically using the present time-accurate 3D finite volume algorithm using body-fitted curvilinear grids, where the unresolved residual stress tensor appearing in the
resolved momentum equations is simulated by an eddy viscosity based turbulence model. In case of stationary grid under a general curvilinear coordinate system, the relevant momentum equation for the resolved velocity component $U_i$ along $i$ direction and the continuity equation are written as following where the other nomenclature has already been given in the previous subsection.

**Mass Conservation**:

$$\frac{\partial}{\partial x_j} \left( \rho \ddot{U}_i \beta_k^j \right) = 0 \tag{5}$$

**Momentum Conservation**:

$$\frac{\partial (\rho \ddot{U}_i)}{\partial t} \frac{1}{f_J} \frac{\partial}{\partial x_j} \left[ (\rho \ddot{U}_i \dot{U}_k - \rho \ddot{U}_i \ddot{U}_k) - \frac{\mu}{f_J} \left( \frac{\partial \ddot{U}_i}{\partial x_m} B^{m}_j + \frac{\partial \ddot{U}_i}{\partial x_j} B^{m}_i \right) \right] + p \ddot{\beta}_k^j - \tau_{ik} \ddot{\beta}_k^j = 0 \tag{6}$$

The Residual or SubGrid Scale (SGS) stress tensor $\tau_{ik}$ may be expressed as the following:

$$\tau_{ik} = \rho (\ddot{U}_i \dot{U}_k - \ddot{U}_i \ddot{U}_k) \tag{7}$$

and this $\tau_{ik}$ needs to be modeled in order to close the filtered equations for $\ddot{U}_i$. The SGS model proposed by Smagorinsky [26] is the simplest linear eddy viscosity based turbulence model where the residual stress ($\tau_{ik}$) in Eq. 7 is related to the filtered rate of strain as following where the first and second term of the right hand side are the anisotropic and isotropic components respectively:

$$\tau_{ik} = \frac{\mu_{sgs}}{f_J} \left( \frac{\partial \ddot{U}_i}{\partial x_m} B^{m}_k + \frac{\partial \ddot{U}_i}{\partial x_k} B^{m}_i \right) - \frac{2}{3} \delta_{ik} k_{sgs} \tag{8}$$

$\mu_{sgs}$ is the eddy viscosity and $k_{sgs}$ is the turbulence energy of the residual motions. The isotropic part is usually absorbed in the resolved pressure term and the anisotropic part is clubbed to the viscous diffusion term of Eq.(6) as following:

$$\frac{\partial (\rho \ddot{U}_i)}{\partial t} \frac{1}{f_J} \frac{\partial}{\partial x_j} \left[ (\rho \ddot{U}_i \dot{U}_k - \rho \ddot{U}_i \ddot{U}_k) - \frac{\mu_{sgs}}{f_J} \left( \frac{\partial \ddot{U}_i}{\partial x_m} B^{m}_j + \frac{\partial \ddot{U}_i}{\partial x_j} B^{m}_i \right) \right] + \left( p + \frac{2}{3} k_{sgs} \right) \ddot{\beta}_k^j = 0 \tag{9}$$

The field values $\mu_{sgs}$ and $k_{sgs}$ are computed through the following algebraic expressions based on mixing length hypothesis and used successfully by previous researchers [19, 22].

$$\mu_{sgs} = \rho I^2 S = \rho \left( f_w C_s \Delta \right)^2 S \tag{10}$$

where, $S$ is the magnitude of the characteristic filtered rate of strain, $l$ is the Smagorinsky mixing length, proportional to the filter width $\Delta$, derived typically from the grid resolution as cube root of the volume of the control cell, $C_s$ and $C_k$ are closure coefficients and $f_w = 1 - \exp (y^+ / 25)$ is a suitable Van Driest kind of damping function which allows the mixing length $l$ to vary exponentially from zero at wall to $C_s \Delta$ in the turbulent boundary layer. Eqs. (5, 9) and (10) now form a closed set of equations to be solved for the resolved velocity components $\ddot{U}_i$ and the resolved pressure $p$ as function of space and time. The mean resolved Reynolds stress components may then be evaluated from the time averaging of the correlation of corresponding resolved fluctuating velocity components $\ddot{U}_i$ expressed as the difference between the instantaneous filtered velocity $\ddot{U}_i$ and the time average of $U_i$, obtained over a large number of time cycles after a statistically stationary state is reached.

**Numerical Solution of Finite Volume Equation**

The relevant Navier Stokes equations (Eq. 1 or Eq. 9) are first transformed to corresponding finite volume equations in terms of surface flux balance for each control volume using the Gauss Divergence theorem. An implicit predictor-corrector method based on a pressure-velocity solution strategy is used for numerical solution of the finite volume equation system. Second order accurate Central Difference or higher order low diffusion Upwind schemes are used for spatial discretisation of the convective fluxes whereas the temporal derivatives are discretised using the second order accurate three-level fully implicit scheme. Using the relevant geometric factors, appropriate discretisation schemes and linearisation of the source terms, the flux balance equations to be solved for momentum and
turbulence scalars are expressed in a generalised implicit manner as follows at the predictor step:

\[
(1.5 \phi_p^{n+1} + 0.5 \phi_p^{n-1} - 2 \phi_p^n) \Delta V/\Delta t = \sum A_{nb} \phi_{nb}^{n+1} + SU - A_P \phi_P^{n+1}
\]

(11)

where \( A_P = \sum A_{nb} - SP \); the coefficient \( A_{nb} \) represents the combined effect of convection and diffusion at the six faces of a hexahedral computational cell; \( SU \) and \( SP \) are the components of the linearised source term, \( \Delta V \) is the cell volume and \( \Delta t \) is the time step size. In the corrector step, the continuity equation is also transformed to a similar linearised equation for pressure correction in the form of Eq.11. The corrections for pressure and velocity field obtained are added to the pressure and the momentum-satisfying velocities at the cell centers and cell faces, obtained at the predictor step. The derivation of Eq.11 and the decoupled iterative procedure to handle the pressure-velocity link are given in details elsewhere [14, 15]. The system of linearised equations (Eq. 11) are solved at each outer iteration level using the strongly implicit procedure of Stone [28].

**Results and Discussion**

**Laminar Flow Past a Bluff Body Mounted on the Lower Wall of a Plane Channel**

This test case is chosen to demonstrate the predictive capability of the present URANS algorithm for non-periodic time-dependent laminar flows. In order to study unsteady flow separation on a bluff body mounted on the lower wall of a plane channel, flow visualization experiments have been conducted [1] using Laser Induced Fluoroscence (LIF) technique in a special water tunnel consisting of a two-component glass channel where the required flow velocity-time variation (Fig.1(a)) at the tunnel inlet is maintained by a servomotor system. The test plane is illuminated by a 2D Laser sheet and the image of the Fluorescent Sodium dye introduced on the body surface, is captured by a CCD camera at different instants of time during the piston motion. The URANS computation domain is bounded by the horizontal channel top wall, the test body geometry for the channel lower wall, an inflow and outflow plane at a distance of 8C on either side, where \( C \) is the channel height. Close view of the H-Grid (191x81) near the bluff body, used for computation is shown in Fig. 1(b). The algorithm uses central difference scheme for convective fluxes and second order time discretisation with time step size (\( \Delta \)) of 0.002 units. Fig.1(c) compares the computed instantaneous streamlines to the flow visualization pictures at three different time instants starting from rest. Reasonably good agreement is observed between the computation and measurement for the instant and the location of the inception of unsteady flow separation and also for the size of the separation bubble growing with time.

**Turbulent Flow Past a Symmetric Aerofoil at High Angles of Attack**

At angles of attack near or beyond the stall angle of an aerofoil, the flow invariably separates from the suction surface leading to periodic vortex shedding and eventually generates a wake consisting of multiple vortical flow structures which can be captured through solution of unsteady flow equations only. A 2-block O-grid (Fig. 2(a)), consisting of 320 x 100 control volumes, has been employed with the far field placed at a radius of 30C and the minimum wall normal distance is maintained at around 8 \( \times 10^{-6} \) C, where \( C \) is the aerofoil chord length. The third order accurate QUICK [11] scheme and second order accurate three-level implicit scheme have been used for the spatial and temporal discretisation of convective fluxes. Fig. 2(b) shows the typical flow boundary conditions, where the farfield is treated either as an inflow or an outflow depending on the sign of the convective flux on the relevant face. Convective boundary condition is used for outflow boundaries. In the multiblock computation environment, every block-interface (cut) boundary is provided with one overlapping control volume on either side of the cut for appropriate transfer of the solution from the neighbouring block. Fig. 3 shows the typical instantaneous particle traces and vorticity contours for three different angles of attack, computed using the S-A turbulence model [27]. The figures clearly show the distinct difference in the vortical structure of the wake flow as the angle of attack changes from 25° to 90°. At \( \alpha = 25° \), a small trailing edge vortex is shed from the suction surface of the aerofoil and as angle of attack \( (\alpha) \) increases to 50°, a large clockwise vortex is formed covering a large part of the suction surface indicating significant enhancement of the lift coefficient. Finally at \( \alpha = 90° \), the aerofoil behaves almost like a plate normal to the flow. A very large anticlockwise vortex covering almost the full blockage area behind the aerofoil, is shed from the aerofoil surface, eventually forming a typical vortex street further downstream in the wake region. The instantaneous aerodynamic coefficients are calculated from the integration of the
tangential wall shear stresses and the wall-normal pressure forces over the whole aerofoil surface. Table-1 shows the sensitivity of the turbulence model on the computed mean aerodynamic coefficients and the Strouhal number at $\alpha = 90^\circ$. In spite of good qualitative agreement, significant scatter is observed in both computation results as well as in the measurement data reported by different researchers [8, 24,25].

Turbulent Flow Past a Pitching NACA 0012 Aerofoil

Flow past a pitching NACA0012 aerofoil is another example of a periodic unsteady flow caused by the sinusoidal motion of the aerofoil surface. This test case [23] validates the accuracy and adequacy of the present URANS algorithm, specially with moving boundary conditions, against the measurement data of McAlister et al. [17]. The aerofoil is subjected to a pitching motion about the quarter-chord point, defined as $\alpha = \alpha_0 + 0.5(\alpha_1 - \alpha_0)(1 - \cos kt)$ where $\alpha_0 = 5^\circ$, $\alpha_1=25^\circ$ and $k (=\omega C/U_\infty) = 0.15$ is the reduced frequency of the oscillatory motion where $\omega$ is the physical frequency. The same numerical grid and convective flux discretisation scheme, used for stationary aerofoil test case described in the previous subsection, are employed in the pitching aerofoil flow situation. Figs.4 (a) and (b) show the instantaneous particle traces and the history of the surface pressure on the aerofoil at four different instants of the pitching cycle. The prediction of the upper surface pressure agrees reasonably well with the measurement data [17] for different angles of attack encountered during the up and down stroke of the pitching motion of the aerofoil. At maximum value of $\alpha = 25^\circ$, whole of the suction surface is observed to be covered by a large clockwise vortex with a small counter-clockwise vortex near the trailing edge which, during the downward motion, eventually pulls the large hysteresis vortex towards the trailing edge and hence reduces the vortex strength on the suction surface. As the value of $\alpha$ decreases during the downward motion, the large single vortex breaks into multiple small vortices, leading to sudden reduction of the suction pressure on the upper surface and hence to drastic loss of lift. Figs.4(c) and (d) show the dynamic hysteresis loops for the lift and drag coefficients computed using different turbulence models. Results using the SA and SST turbulence models are observed to be in the better agreement with the measurement data. The double peaking of $C_l$ during the start of the return downward stroke of the aerofoil, is reasonably captured by all the models. However quantitative disagreement observed at some regions, may perhaps be attributed to the uncertainties and inadequacies of the eddy-viscosity based turbulence models used.

Turbulent Flow Past an Aerostat Balloon

The main objective of selecting this steady flow problem is to demonstrate the capability of the multiblock parallel version of the present URANS algorithm even for prediction of steady three-dimensional turbulent flow past any complex arbitrary shaped geometry. This problem is about prediction of a high Re ($Re=1.5x10^7$) turbulent flow around an Aerostat Balloon configuration consisting of an axisymmetric hull generated from a smooth aerodynamic shape along with three fins of aerofoil (NACA0018) cross section, attached near the tail end of the hull as control surfaces. The computation results [16] are validated against corresponding Panel code results [20] and also the measurement data obtained from wind tunnel tests on a 1/7th scaled model [7]. Circumferentially stacked 2D grid is used in the domain outside the fin region whereas the grid (Fig.5(a)) on the fin-surface are laid out separately using a purely algebraic procedure. QUICK scheme [11] with deferred correction approach [9] and $k - \varepsilon$ turbulence model with standard wall function have been used for the flow computations. The computation domain

Table-1 : Mean Aerodynamic Coefficients and Strouhal Number for NACA0012 at $\alpha = 90^\circ$

<table>
<thead>
<tr>
<th></th>
<th>Present Computation</th>
<th>Computation Shur et.al</th>
<th>Measurement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lift, $C_l$</td>
<td>0.060</td>
<td>0.053</td>
<td>0.10</td>
</tr>
<tr>
<td>Drag, $C_d$</td>
<td>2.741</td>
<td>3.155</td>
<td>2.84</td>
</tr>
<tr>
<td>Strouhal Number</td>
<td>0.098</td>
<td>0.107</td>
<td>0.07</td>
</tr>
</tbody>
</table>
covering 300 x 82 x 92 control volumes has been decomposed into eighteen number of blocks computed in parallel using six Pentium processors of NAL Parallel machine Flosolver Mk5, each covering three consecutive blocks along circumferential direction. Fig.5(b) shows the pressure distribution along the top surface generator line of the hull at three different angles of attack (α). The comparison clearly shows reasonable agreement of the measurement data to the present RANS solution and also the expected large discrepancies between measurement data and the Panel code results at high angles of attack.

**Turbulent Flow Past a Circular Cylinder at Re=3900**

This is a classic example of an unsteady three-dimensional flow consisting of separation, reattachment, three dimensionalities, free shear layer instabilities, laminar to turbulent transition and vortex shedding in the cylinder wake. Since the URANS approach with eddy viscosity-based turbulence model are reported [2, 3, 6] to be inaccurate and unreliable for transitional flows, this test flow is identified as the ideal candidate for validation of the LES methodology. A four block cylindrical polar grid is used to cover the annular computation domain formed by a unit diameter cylinder, the far field circular boundary at a radius of 20 units and a spanwise length of 2π units. The computation domain covering 120 x 145 x 30 control volumes has been decomposed into four blocks to be computed in parallel using four processors of an SGI Altix machine, each processor covering one quarter segment of the cylinder along the circumferential direction (Fig. 6(a)). No slip condition is used on the wall surface, whereas the far field boundaries are treated as inflow or outflow depending on the sign of the local mass flux. The spanwise end planes are treated as periodic boundaries. The computation using a timestep size of 0.05 units, takes about 6 clock hours to reach a statistically stationary state. Grids are equispaced along circumferential and spanwise direction, and stretched radially near the cylinder wall for fine resolution of the boundary layer. A low sub-critical Reynolds number of 3900 is chosen for which the measurement data show that the flow separation on cylinder wall is laminar and the transition to turbulence takes place in the free shear layers. Once a statistically stationary state is reached, computation is continued further and the value of the flow variables are averaged over number of time steps required to cover at least 30 vortex shedding cycles and also averaged over the spanwise direction for comparison to measurement data [12, 13, 21] on the midspan plane (z = 0). Fig. 6(b) shows the computed isosurfaces of the instantaneous vorticity magnitude (ω/D/U∞) in the cylinder wake for fourteen different values ranging from 0.5 to 10. The figure clearly demonstrates the two free shear layers formed following the flow separation from the cylinder surface and also the complex three-dimensional wake structure developed due to the interaction between the shear layers and the primary Karman vortex street behind the cylinder. Further, two isosurfaces of streamwise components of vorticity (ωx/D/U∞ = ±0.7) of equal magnitude with opposite signs, represented by two different colours in Fig.6 (c), demonstrate the flow structures consisting of streamwise vortices of alternating signs along the spanwise direction, typically formed in three-dimensional flows over a finite span. Figs. 6(d) and (e) show reasonable agreement between the measurement data and the present 3D LES computation results (both with and without SGS model) for the circumferential variation of the average surface pressure and also for the variation of the mean streamwise velocity along the wake centerline. On the other hand, the 2D LES with no model clearly fails to predict the attached recirculation region behind the cylinder, observed in measurement data and yields inaccurate results for the surface pressure variation as well. Figs. 6(f) to 6(i) compare the present LES predictions and the corresponding measurement data for the transverse profiles of the mean streamwise velocity component and three different components of the total resolved Reynolds stress at the longitudinal station x/D=1.54. Reasonable agreement is observed for all the flow quantities between the measurement data and the present LES computation results. Better agreement observed between measurement data and LES results with SGS model indicates the significant effect of the SGS model to capture the momentum and energy transport mechanism between the resolved large scale structures and the small subgrid scale eddies for the chosen grid size, even at moderately low Re value of 3900.

**Concluding Remarks**

A fully implicit second order accurate pressure-based Navier Stokes solver in generalised body-fitted non-orthogonal coordinate system, has been developed at the CTFD Division, NAL, Bangalore, for time-accurate calculation of incompressible turbulent flow in or around arbitrary shaped configurations. The simulation of turbulence may be carried out using either URANS or LES approach to be chosen by the user. The algorithm is also parallelised efficiently using the domain decomposition method, coupled to multiblock structured grid for handling complex configuration. The code validation studies in laminar situations, have demonstrated the accuracy and
adequacy of the spatial and temporal discretisation schemes used, the handling of moving boundaries in an inertial frame of reference and the proper implementation of parallelisation of the algorithm in a multiblock structured grid environment. For turbulent flows, the URANS approach coupled to variety of linear eddy viscosity based turbulence models with special near wall treatments are found to be reasonably accurate for prediction of turbulent boundary layer flows with mild flow separation under moderate adverse pressure gradients. The superior performance of LES approach for more realistic and accurate simulation of turbulence physics for complex flows in presence of transition, flow unsteadiness, strong effects of curvature or rotation, has been established through the example of flow past a circular cylinder. Work is in progress to incorporate Non-Linear Eddy Viscosity based models and Dynamic SGS models for more accurate URANS and LES computations respectively. Capabilities of the code are being further enhanced to handle multiphysics problems in future incorporating models for multiphase flow, free-surface flow and reacting flows.

Acknowledgements

The authors wish to thank the Directors, Aerial Development Research and Development Establishment (ADRDE) Agra, NSTL, and the Hydrodynamics Panel of the Naval Research Board, India for the financial support provided to this development work through several research projects sponsored to NAL, Bangalore. The authors also wish to thank the Director, NAL for his kind permission to publish this paper.

References


Fig. 1 Unsteady laminar flow past a bluff body mounted on the lower wall of a channel
Fig. 2 Grid and boundary condition used for aerofoil flow computation

Fig. 3 Instantaneous particle traces and vorticity contours for flow past NACA0012 aerofoil at different angles of attack (S-A turbulence model at Re = 106)
Fig. 4 Turbulent flow past a pitching NACA0012 aerofoil \((Re = 10^6, k = 0.15 \text{ and } \Delta t = 0.05)\)
Fig. 5 Turbulent flow around aerostat ballon with fins (Re = 1.5 \times 10^7)
Fig. 6 Large eddy simulation of turbulent flow past a circular cylinder at $R = 3900$