7. ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2013-162**

11-13 September 2013 - METU, Ankara TURKEY

PARALLEL IMPLEMENTATION OF DETACHED EDDY SIMULATION ON 2D HYBRID GRIDS

Ozcan Yirtici* and Ismail H. Tuncer†
Middle East Technical University
Ankara, Turkey

ABSTRACT

In this study, Detached Eddy Simulation turbulence model is studied in turbulent flow solutions over a flat plate and flow over single element airfoils in high Reynolds numbers. For this aim, DES turbulence model is implemented into the parallel, viscous, hybrid grid flow solver ,Hype2D, . The obtained results are compared with the numerical and experimental studies for subsonic flows. It is seen that Spalart-Allmaras and DES turbulence models agree well with each others at low angle-of-attacks.

INTRODUCTION

Nowadays, the numerical solutions of engineering applications are required to solve turbulent flows in high Reynolds numbers. Since these turbulent flows contain large and small eddies together, there are four main methods for predicting the effect of turbulence with advantages and disadvantages in terms of computational cost and simulation quality. These are Direct Numerical Simulation (DNS), Large Eddy Simulation (LES), Reynolds Averaged Navier-Stokes (RANS) and Hybrid LES/RANS methods. In DNS, all scales of eddies are simulated without any turbulence modeling. It requires very dense grid therefore even with today's computer power the applicability of this method is impossible for engineering applications. In LES, the governing equations are spatially filtered to separate the large scales of eddies from the small scale eddies depending on the mesh size. While large scale eddies are resolved like DNS, small scale eddies are modeled by sub-grid scale (SGS) models. The main challenge to practical use of LES on engineering applications is fine mesh resolution which is comparable with DNS near the wall boundaries. In RANS the mean flow quantities are solved. Although, obtaining solution with this method consumes less time and computer power than the others, it could not predict the separation and the reattachment correctly. The main purpose of hybrid LES/RANS methods are to combine RANS turbulence model with LES for obtaining more accurate results in affordable computational cost. While RANS is used near the walls, the remaining regions are solved by LES. The resolution of mesh has an important place in hybrid methods, since the grid density is much denser than RANS grids. It requires RANS mesh resolution near walls and isotropic mesh cells elsewhere. Therefore, even in 2D flow simulations parallel implementation is needed for the implementation of hybrid models. In his review paper, Spalart [Spalart PR. , 2009] summarized the history, current status and perspectives of DES.

In this study, a well known hybrid model, Detached Eddy Simulation- DES, is implemented in an inhouse flow solver, Hype2d. Hype2d is a 2D finite volume based viscous flow solver on hybrid grids. In order to accurately resolve the boundary layers in wall bounded turbulent flows, quadrilateral grid cells are employed in the boundary layer regions normal to solid surfaces while the rest of the domain is discretized by triangular cells. The computation time requirement, which is a significant deficiency in the implementation of the DES model, is alleviated by performing the computations in parallel computing environment. Turbulent flow solutions over a flat plate and single element airfoils are presented and the results obtained are compared to the numerical

*GRA in Aerospace Engineering Department, Email: oyirtici@ae.metu.edu.tr
†Prof. in Aerospace Engineering Department, Email: tuncer@ae.metu.edu.tr

and experimental studies. It is seen that Spalart-Allmaras and DES turbulence models agree well with each other for attached flows.

METHOD

Detached Eddy Simulation is based on Spalart-Allmaras turbulence model. In the DES model, distance to the closest wall ,d, is replaced with a new length scale $\tilde{d}$ away from solid surfaces. model:

$$\tilde{d} = MIN(d, C_{DES}\Delta) \tag{1}$$

where C_{DES} ,being the DES coefficient, is equal to 0.65 for homogeneous turbulence. Close to the wall ,where $d < \tilde{d}$, the model employs as Spalart-Allmaras turbulence model and away from the wall ,where $d > \tilde{d}$, the model turns into LES with smagorinsky SGS model. Δ represents the cell size and plays an important role in the model due to the fact that switching from RANS to LES depends on it. There are various implemetations:

$$\Delta = MAX(\Delta_x, \Delta_y) \tag{2}$$
$$\Delta = (\Delta_x \Delta_y)^{\frac{1}{2}} \tag{3}$$
$$\Delta = (\frac{\Delta_x^2 + \Delta_y^2}{2})^{\frac{1}{2}} \tag{4}$$

In this study, the simple expression given in Eqn. 2 is employed [Yirtici, 2012].

Parallel Algorithm

Turbulent flow simulations with DES turbulence model require considerable amount of computer time due to grid density. In the present study computations are preformed in parallel based on domain decomposition. Parallel Virtual Machine (PVM) libraries are used to pass information between the processors, and METIS is used to partition the grid into sub-domains. The parallel algorithm is based on the master-worker paradigm. While master code initializes the solution, organizes the tasks of workers and gather the results, worker code performs the time integration of the N-S equations and exchanges the inter-domain boundary conditions. Further details of the parallel implementation is given in [Yirtici, 2012].

RESULTS AND DISCUSSION

First of all, parallel validation of implemented DES turbulence model is done at turbulent flow past a flat plate. Then, the effect of the first grid distance on solutions and parallel performance of the present solver is tested on transonic flow solutions over RAE2822 airfoil for steady state flow condition. Lastly, performance of Spalart-Allmaras and DES turbulence models of the present solver are analyzed at the flow over Aerospatiale and NACA 0015 airfoils. Obtained results are compared against the available data and results of Fluent solver.

Validation Case : Turbulent Flow Over a Flat Plate

In this validation case, the turbulent flow over a flat plate with free stream conditions Mach 0.2 and Reynolds Number $5x10^6$ is studied. The mesh used in this simulation is shown in Figure 1.

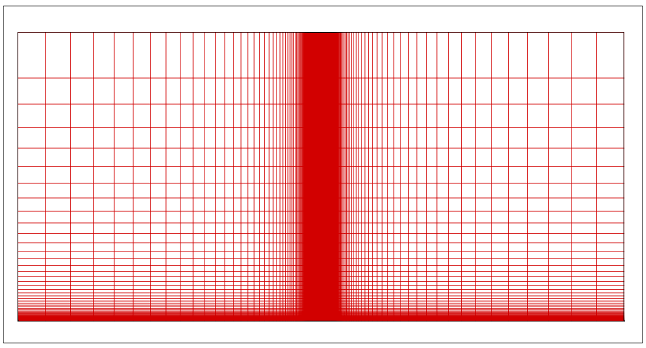

Figure 1: Mesh used in the Flat Plate Simulation

A rectangular solution domain is discretized and the thickness of the first layer adjacent to the flat plate is $4.5E-7$ unit. The mesh contains of 23684 nodes and 23370 cells. Figure 2 shows the Universal Velocity Distribution. As it can be seen from Figure 2, the computational velocity profile is in agreement with the Universal Velocity Distribution in almost every region of the turbulent boundary layer. Inside viscous sublayer, buffer zone and log layer zone the computed velocity profile matches with the analytical and semi-emprical distribution. This figure shows that implementation of DES turbulence model successfully simulates the turbulent boundary layer development over a flat plate.

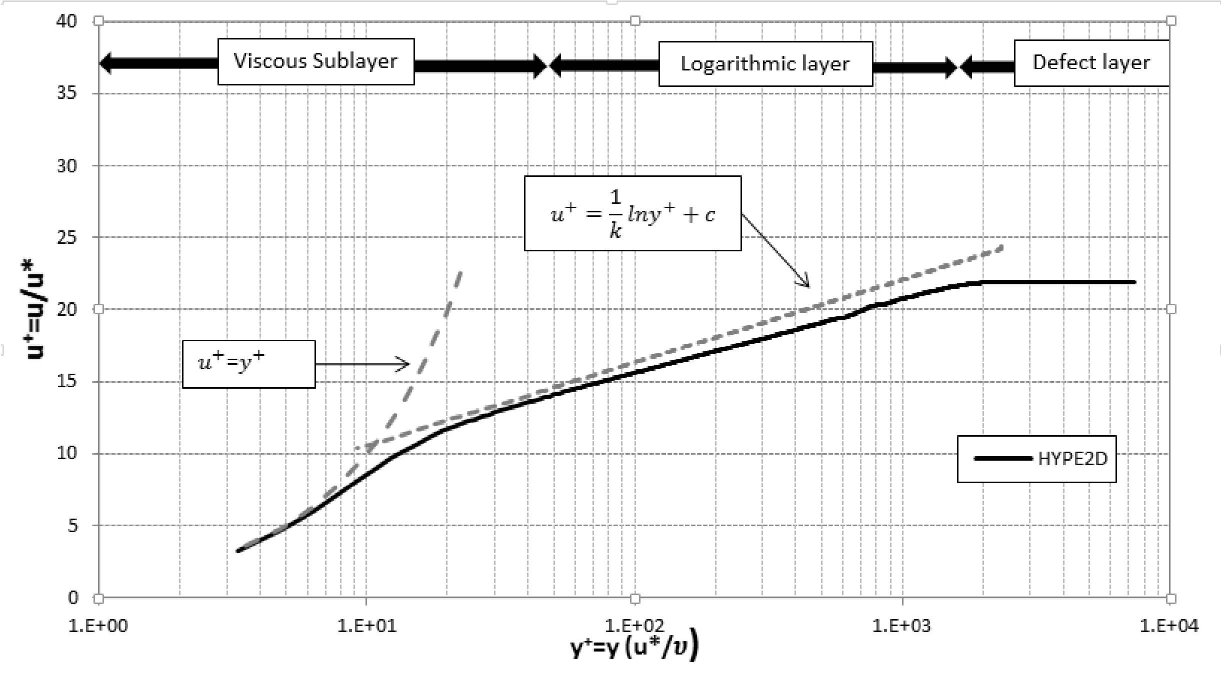

Figure 2: Non-Dimensional Velocity Profile for Turbulent Flow Over a Flat Plate and Identification of Different Regions within the Turbulent Boundary Layer

Transonic Flow Solutions over RAE2822 Airfoil for Steady-State Flow Condition

In this case, transonic, turbulent flow over RAE2822 airfoil with free stream conditions Mach 0.73, Reynolds Number $6.5x10^6$ and $\alpha = 2.8°$ is studied. The effect of first grid distance is investigated for three different y^+ values. Node number over the airfoil surface is kept constant and features of these grids are given in Table 1.

Table 1: Y^+ Values of Used Grids for Turbulent Flow over RAE2822 Airfoil

Mesh	# of nodes over airfoil surfaces	# of nodes	# of cells	y+
Mesh-$y^+ = 0.95$	440	24026	38058	$y^+ \approx 0.95$
Mesh-$y^+ = 0.5$	440	24094	38194	$y^+ \approx 0.5$
Mesh-$y^+ = 0.1$	440	26363	40652	$y^+ \approx 0.1$

Figure 3 shows close up views of Mesh-$y^+ = 0.5$ solution domain. All of the grids have the same topology. The chord length of RAE2822 airfoil is 1 unit while farfield boundary is taken as 11 units. The distance from the first layer of the boundary layer to the closest wall is $2x10^{-6}$ units.

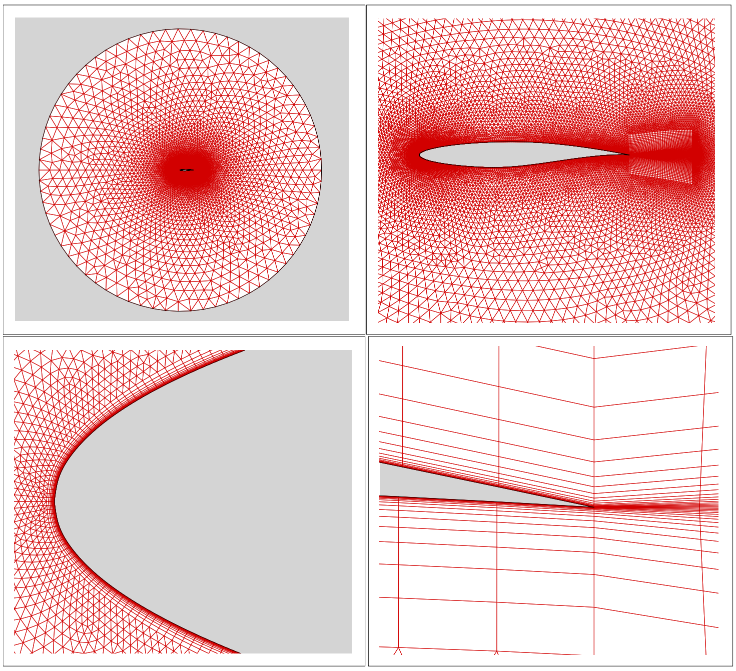

Figure 3: Detailed view of Mesh-$y^+ = 0.5$

Residual histories are presented in Figure 4. At least, a reduction of four order of magnitude in density and five order of magnitude in the turbulent viscosity residuals are achieved.
The obtained pressure and skin friction distributions are compared both with the experimental data which is

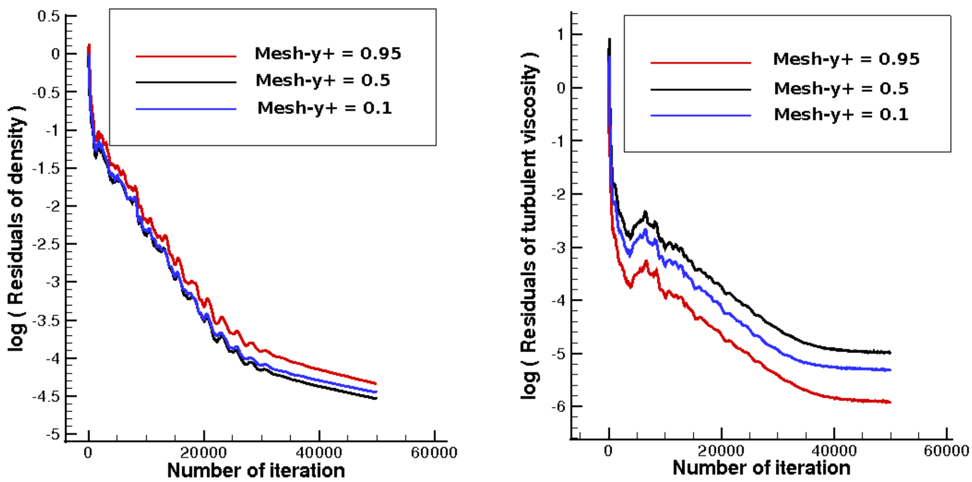

Figure 4: Converge History for Different y^+ Values

found in AGARD Advisory Report no. 138 Thibert et al. [1979] and numerical solution of Chit et al. [2009]. The solution is assumed to reach steady state when the reduction of density residual is less than four order of magnitude in turbulent flow cases. The pressure and the skin friction distributions over airfoil for different y^+ value are demonstrated in Figure 5. It is seen that different y^+ values do not make any change on the solution. This shows that $y^+ \leq 1$ value is enough to obtain turbulent flow solutions.

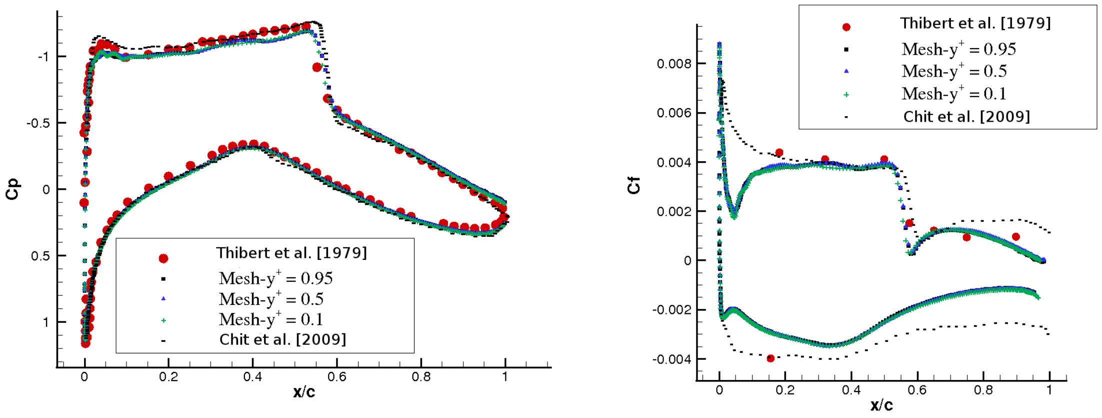

Figure 5: Pressure and Skin Friction Coefficient Distrubition

The parallel performance of present code is investigated with the computational grid of Mesh-$y^+ = 0.5$ for 30000 iterations. Information about Mesh-$y^+ = 0.5$ solution domain can be found in Table 1 and detailed view of this solution domain can be seen in Figure 3.

Performance of parallel computing is investigated with respect to CPU time. The speed-up and the efficiency of the present solver are demonstrated in Figure 6. As expected speed-up increases linearly with respect to processors numbers however after using more than eight processors, it is observed that speed up slope is decreased and flattened out at a level of seven speed-up factor.The efficiency of the present solver is decreased as the processors numbers increases. This outcome is expected since the solution domain has low grid density. The major portion of its execution time is used up by communication for data exchange rather than computation. However, for high density grids where large number of grid points is required ,3D solutions, parallel solution is expected to provide higher speed-up and efficiency values by using large number of processors.

As the processors numbers increases, much more time is spent for communication between computer hosts in the network. Therefore eight processors will be used for parallel solutions of Spalart-Allmaras and DES turbulence models in present solver to reduce communication time loses.

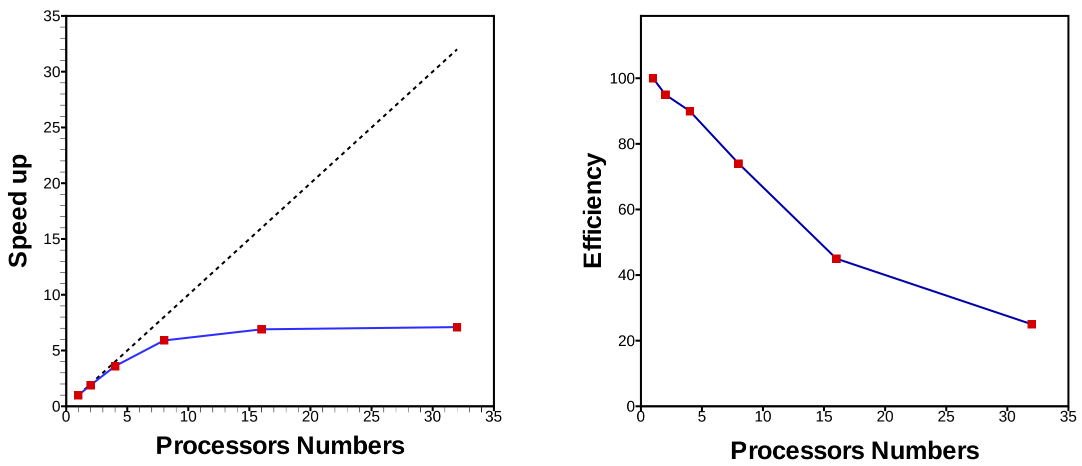

Figure 6: Speed up Factor and Efficiency for the Mesh-$y^+ = 0.5$

Flow Solutions Over Aerospatiale Airfoil

In this case, two dimensional low subsonic flow over Aerospatiale ,A-airfoil, airfoil with free stream conditions Mach 0.15 Reynolds Number $2.0x10^6$ and $\alpha = 7.2°$ is studied. For unsteady flow solutions, after the computations start from free stream solution and marches in time with local cell time stepping till density residual drop four order magnitude, global time stepping is used. The chord length of the solution domain is taken as 1 unit while the farfield boundary is located 11 chord out from Aerospatiale airfoil. The distance from the closest wall to the first boundary layer is $1x10^{-5}$ units. Detailed view of the solution domain is given in Figure 7.

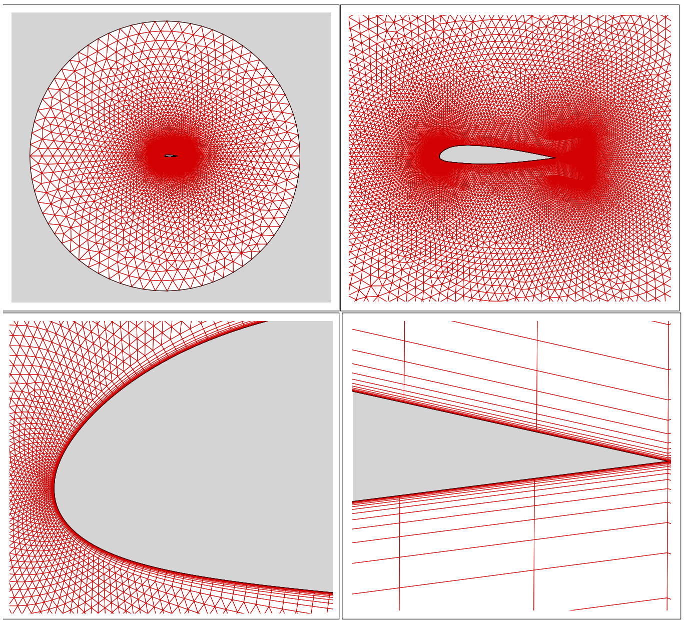

Figure 7: Detailed View of Solution Domain for Aerospatiale Airfoil

Residual convergence histories can be seen in Figure 8. After 60000 iterations, Spalart-Allmaras and DES turbulence models converge to six order magnitude drop in density residual but while Spalart-Allmaras converge to 5.2 order magnitude drop DES converge to 4.8 order magnitude drop in turbulent viscosity residual.

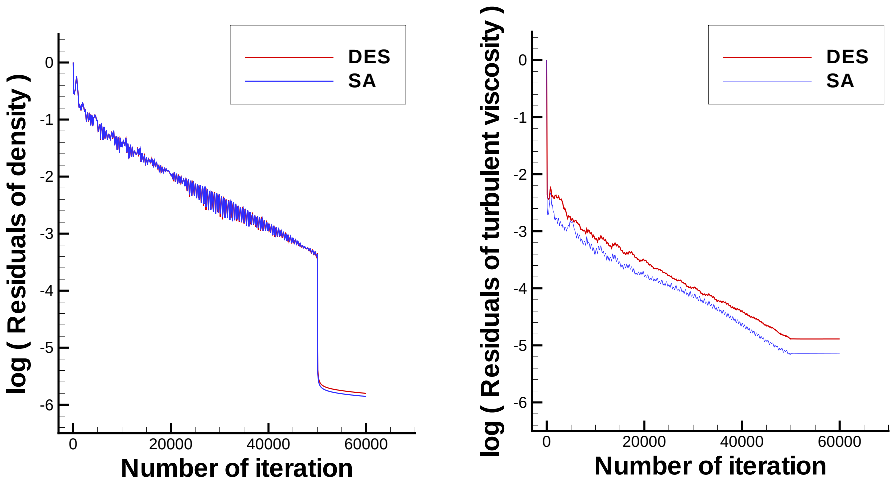

Figure 8: Converge History of Residues for Aerospatiale Airfoil

The pressure coefficient distribution over Aerospatiale airfoil is given in Figure 9 for Spalart-Allmaras and DES turbulence models. It is clearly seen that, at moderate angle of attack, Hype2D-SA and Hype2D-DES gives the same pressure distribution with first order accurate Fluent solver. When compared to experimental data, it is seen that compressible first order solutions of our inhouse and Fluent solvers can not catch the suction pressure peak. Since the same solution domain is used both in Fluent and present solvers, the under prediction of the suction peak pressure of the leading edge is attributed to the first order accurate solutions. Also fully turbulent assumption may contribute to this.

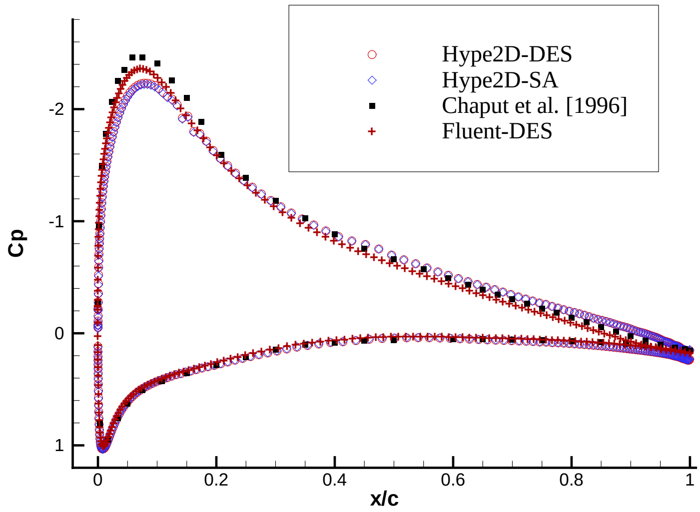

Figure 9: Pressure Coefficient Distrubition over Aerospatiale Airfoil

Flow Solutions Over NACA0015 Airfoil at High Angle of Attack

Turbulent, low subsonic flows over NACA0015 airfoil are first studied by both Spalart-Allmaras and DES turbulence models. The flows are computed for $M = 0.1235$, $Re = 1.5E + 6$ and $\alpha = 12°$. The chord length of the airfoil is taken as 1 unit and the thickness of the first grid layer next to the wall is set to $1x10^{-5}$ units (Figure 10). The hybrid grid consists of 38460 cells with 440 nodes over the airfoil surface. Numerical solutions are initially carried out in the steady state mode with local time stepping technique in order to accelerate the numerical solution. Then, the time-accurate, unsteady flow solution is obtained by employing a single global time step for all the cells.

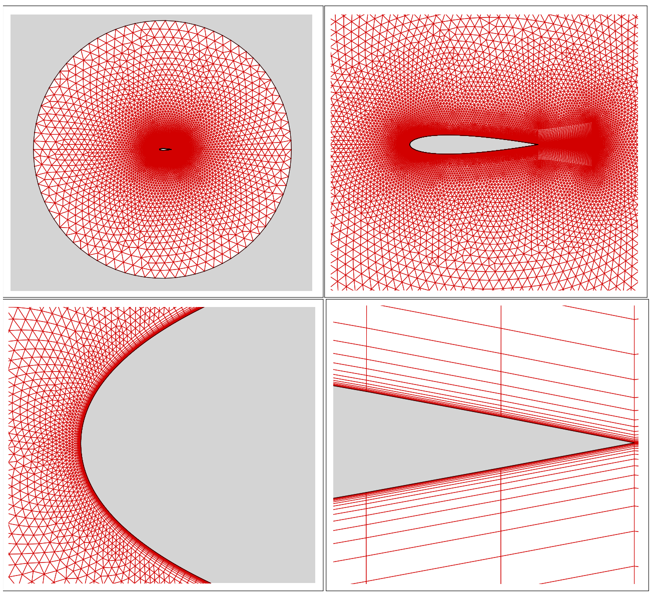

Figure 10: Detailed View of Solution Domain for NACA0015 Airfoil

Figure 11 shows the convergence histories. It is observed that the $L2$ norm of residuals drop more than 4 orders of magnitude along the solution steps with local and constant time stepping, and the lift coefficients converge to the constant values.

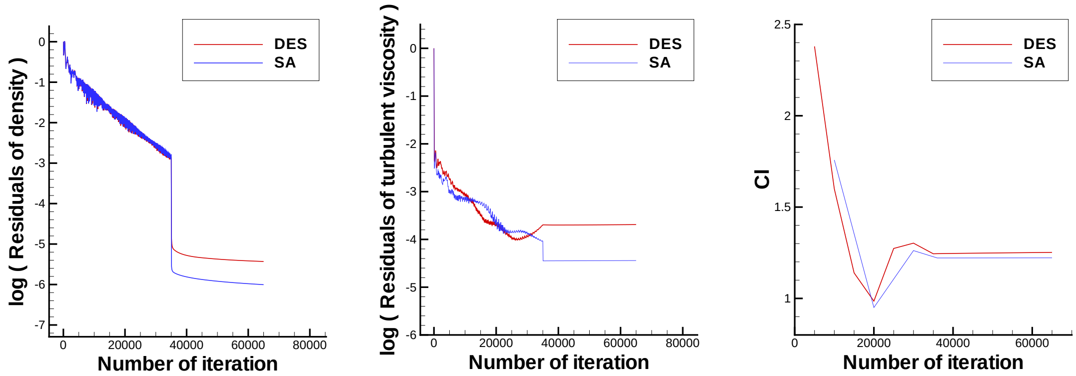

Figure 11: Converge Histories for NACA0015 Airfoil

Table 2: Aerodynamic Coefficients for NACA 0015 Airfoil

Model	C_L	C_D
Hype2D-DES	1.252	0.074
Hype2D-SA	1.222	0.072
Fluent-DES	1.284	0.065
Fluent-SA	1.274	0.058
DES Tu et al. [Tu et al. , 2009]	1.075	0.020
Exp [Sheldahl and Klimas , 1981]	1.132	0.017

The lift and drag coefficients are compared with the experimental data [Sheldahl and Klimas , 1981], the numerical result of Tu et al. [Tu et al. , 2009], and the Fluent solutions in Table 2. It is seen that Hype2D and Fluent predictions are significantly higher than the experimental data, which may be attributed to the first order accuracy of the solutions. In addition, DES solutions predict slighly higher values in comparison to the the SA solutions. DES solution of Tu et al., which has a second order accuracy, agree better with the experimental data.

The distributions of Mach number and the turbulent viscosity ratio fields are given in Figure 12. Although the Mach number distributions in DES and SA solutions are similar, the turbulent viscosity ratio fields are quite different. The DES turbulence model produces sinificanltly less turbulent viscosity than that of the SA turbulence model.

The pressure distributions over the airfoil are compared in Figure 13. As expected, the DES and SA predictions are in good agreement with each others, since the flow is still mostly attached. In comparison to the Fluent solution and the solution of Tu et al. [Tu et al. , 2009], the Hype2D solution has an oscillatory behaviour on the suction side of the airfoil and the suction is predicted slightly higher.

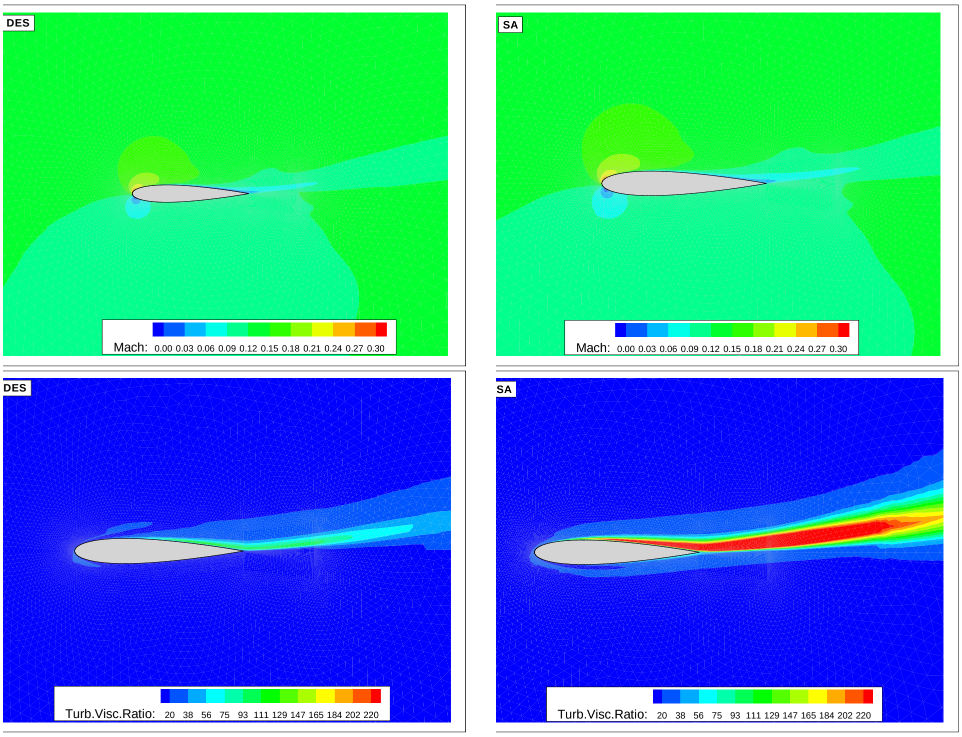

Figure 12: Mach Number and Turbulent Viscosity Ratio Contours for NACA 0015 Airfoil

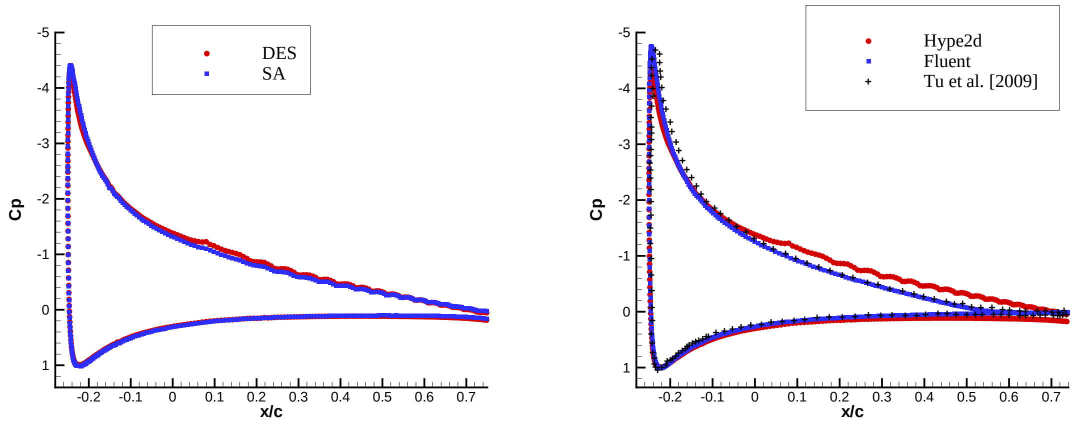

Figure 13: Pressure Coefficient Distribution over NACA 0015 Airfoil

CONCLUSION

In this study, the DES turbulence model is successfully implemented to an in-house flow solver Hype2D. The Navier-Stokes equations are discretized by cell centered Finite Volume Method and solved explicitly by using Runge Kutta time stepping method. The inviscid fluxes are computed by using Roe differencing method. The numerical solution is implemented in a parallel computing environment using domain decomposition algorithm with PVM library routines for inter process communications.

DES turbulence model is studied in turbulent flow solutions over flat plate and flow over single element airfoils in high Reynolds numbers. It is seen that DES and Spalart-Allmaras turbulence models almost produce same turbulence behavior. As seen in the previous Results and Discussions section, the Hype2D solution has an oscillatory behavior on the suction side of the NACA0015 airfoil at high angle-of-attack. In the future, much additional work will be done to complete understanding of this phenomena occurs.

References

Yirtici Ozcan (2012) *Detached Eddy Simulation of Turbulent Flow on 2D Hybrid Grids*, Msc. Thesis, METU

Tu S.,Aliabadi S.,Patel R.,Watts M. (2009) *An Implementation of the Spalart-Allmaras DES Model in an Implicit Unstructured Hybrid Finite Volume/Element Solver for Incompressible Turbulent Flow*, International Journal For Numerical Methods In Fluids, 59: 1051-1062, 2009.

Spalart PR. (2009) *Detached-Eddy Simulation*, Annual Review of Fluid Mechanics, 2009

Chit Ong J., Omar Ashraf A. and Asrar Waqar, (2009)*Reynolds Averaged Navier Stokes Flow Computation of Rae2822 Airfoil Using Gas-kinetic BGK Scheme* , Proceedings of the International Multiconference of Engineers and Computer Scientists, Vol. II, 2009, IMECS 2009, Hong Kong.

W. Haase, E. Chaput et al. (1996), *Validation of CFD Codes and Assessment of Turbulence Models* , European Computational Aerodynamics Research Project, Notes on Numerical Fluid Mechanics, Vol 58, pp. 327-346, 1996

Sheldahl Robert E., Klimas Paul C. (1981) *Aerodynamic Characteristics of Seven Symmetrical Airfoil Sections Through 180-Degree Angle of Attack for Use in Aerodynamic Analysis of Vertical Axis Wind Turbines* , Sandia National Laboratories, Report SAND80-2114, 1981.

Thibert, J.J., Grandjacques, M. and Ohman, L.H. (1979),*Experimental Data Base for Computer Program Assesment* , AGARD, AR-138, May, 1979.