4-28-2016

Application of Wray-Agarwal Turbulence Model for Accurate Numerical Simulation of Flow Past a Three-Dimensional Wing-body

Ning Deng
Washington University in St. Louis

Ramesh K. Agarwal
Washington University in St. Louis

MEMS 500: Independent Study Report

Application of Wray-Agarwal Turbulence Model for Accurate Numerical Simulation of Flow Past a Three-Dimensional Wing-body

Ning Deng

Advisor: Professor Ramesh K. Agarwal

Introduction

Accurate numerical simulation of subsonic/transonic turbulent flow is very important for the design and optimization of a transport aircraft. Computational Fluid Dynamics (CFD) modeling of the subsonic/transonic turbulent flow requires a turbulence model that can predict the flow field with an acceptable accuracy. Currently, most widely used industry standard turbulence models employed in conjunction with the Reynolds-Averaged Navier-Stokes (RANS) equations are the single equation Spalart-Allmaras (SA) model and the two-equation Shear Stress Transport (SST) k-ω model. Recently a new one equation model has been developed by Wray and Agarwal, known in the literature as the Wray-Agarwal (WA) model. The accuracy of this model for computing the transonic flow past a three-dimensional wing-body is evaluated as part of this independent study. Results are compared with those obtained with SA and SST k-omega models.

For a transport aircraft, pressure distribution, lift coefficient and drag coefficient are the key data. In this report, these data are computed for a typical wing/body and are compared using the three different turbulence models – SA, SST k-omega and WA.

Geometry and Computational Domain

The model used is a wing body shown in Figure 1. Some key parameters of the wing body are: mean aerodynamic chord $\bar{c}$ of the wing is 0.6956 m; wing semi-span is 1156.8 mm; reference plan area is 797448 $mm^2$and the Reynolds number based on mean chord is3×10^7.

To save the computational resources, the half-model of wing-body about the symmetry plane is employed in the CFD simulation. A rectangular prism shaped computational domain is employed as shown in Figure 2. The inlet boundary is positioned at $5\bar{c}$ away from the nose of the wing-body; the outlet boundary is positioned at $10\bar{c}$ away from the tail; the upper boundary is positioned at $5\bar{c}$ away from the highest point of the craft; the side boundary is positioned at $5\bar{c}$ away from the wing tip; the lower boundary is positioned at $5\bar{c}$ away from the lowest point of the craft. Thus, the exterior boundary of the computational domain is set at far enough distance from the wing/body so that the drag and lift can be calculated accurately.

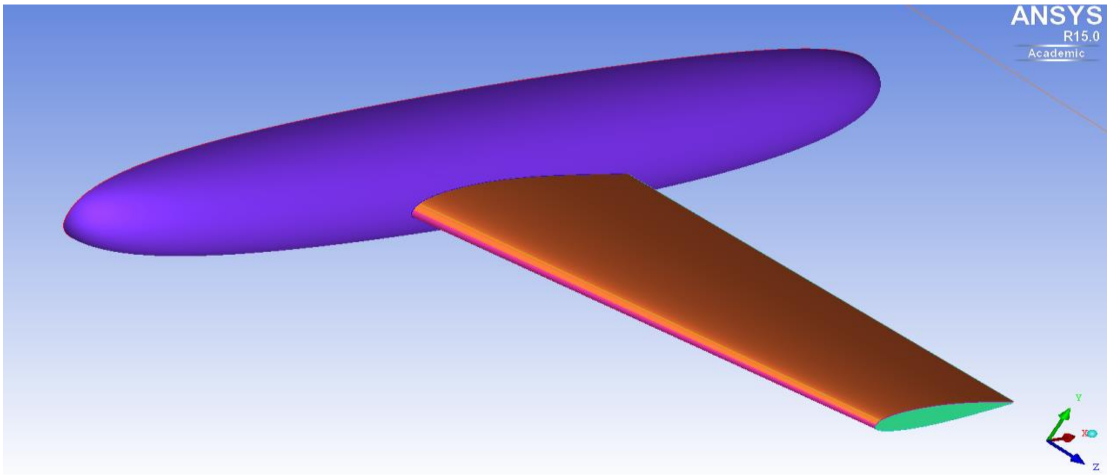


Figure 1: Geometry of the Wing-body

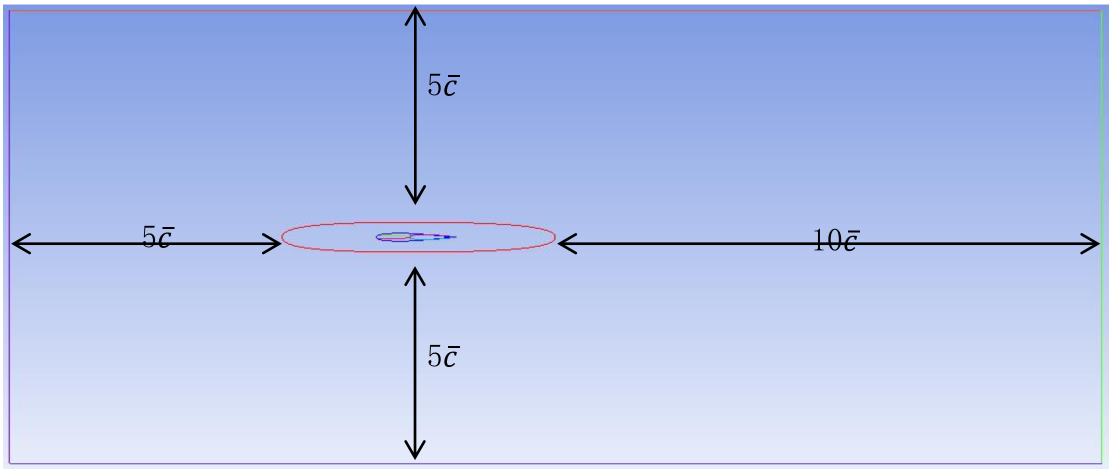


Figure 2: Computational Domain around the Wing-body

Numerical Method

The commercial CFD solver ANSYS FLUENT is employed to perform the CFD analysis. The steady compressible RANS equations are solved in conjunction with the SA, SST k-omega and WA models. Since the flow is subsonic/transonic, density-based solver in FLUENT is employed. In order to speed up the convergence process, the Courant Number of 0.5 is used.

Mesh

A structured mesh on the surface of the wing-body and in the computational domain surrounding it is generated using ANSYS ICEM. The surface grid on the wing-body is shown in Figure 3. A sequence of mesh is used in the simulation to determine a suitable grid for grid independent solutions. The grid independence study is conducted for two cases with alpha = 7°and 16° using the SA and SST k-omega models. Results with a mesh of 3,762,028 hexahedron elements are compared with those obtained with a mesh of 2,016,689 hexahedron elements. The maximum difference between the lift coefficients and drag coefficients (see Table 1) obtained with two meshes is 7.61%. It shows that the grid independence is achieved for both the lift and drag predictions by choosing a finer mesh. For achieving higher computational efficiency during the computational process, the mesh with fewer elements is initially adopted.

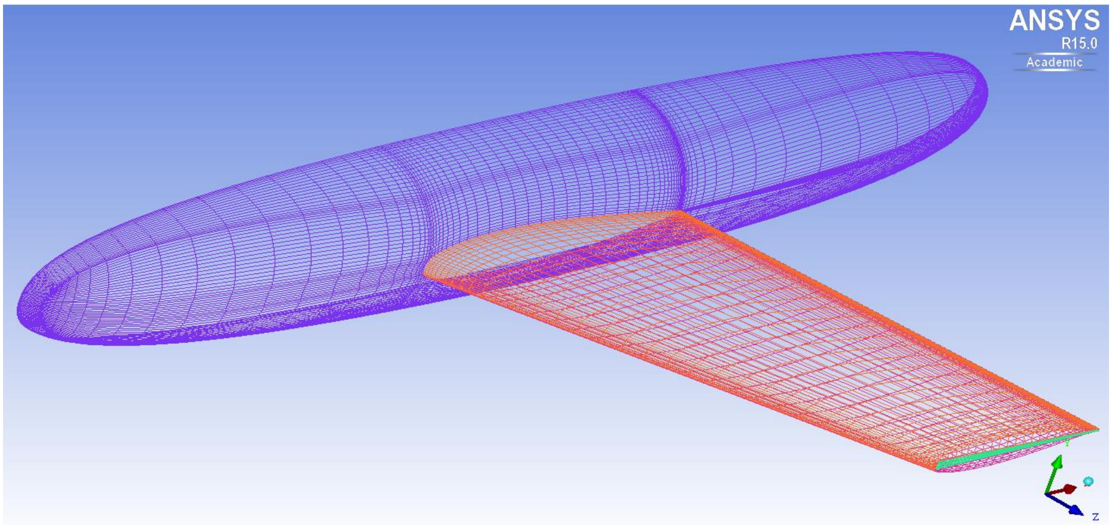


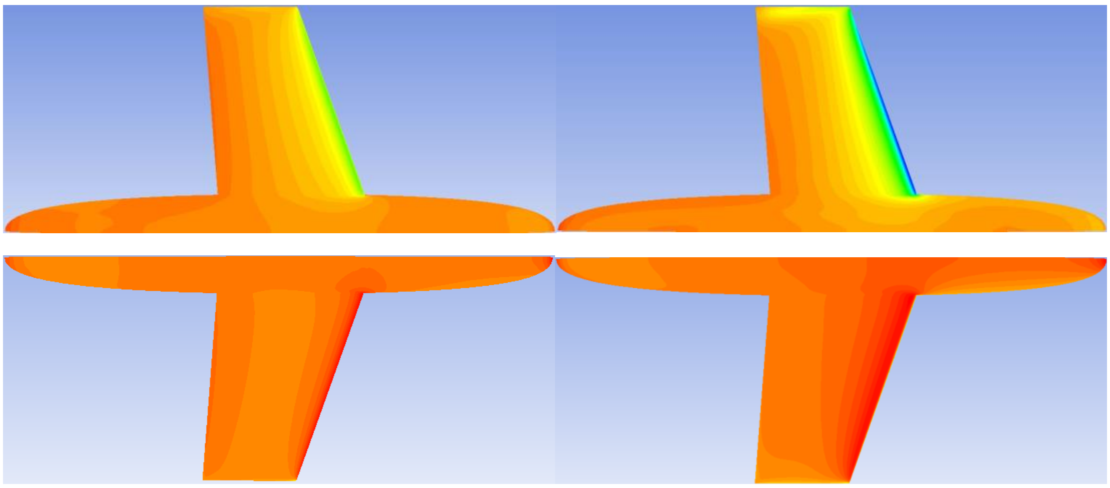
Figure 3: Surface Grid on the Wing-body

Boundary Conditions

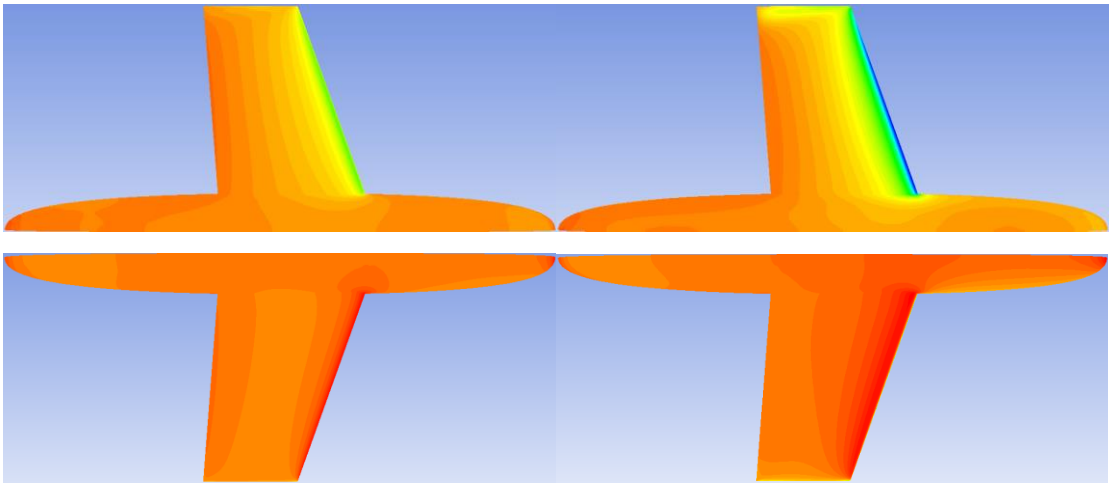
Boundary conditions are set as follows: A pressure far-filed boundary condition is employed at the outer boundary of the computational domain. A symmetry boundary condition is used for the symmetry plane of the wing/body. A no-slip wall boundary condition is used for the wing-body surface. All the far-fields planes except for the symmetric plane are set as pressure far-field with temperature of 114K and pressure of 295000 Pa, and Mach number Ma = 0.175. The angle of attack α is set at 7° and 16°.

Results

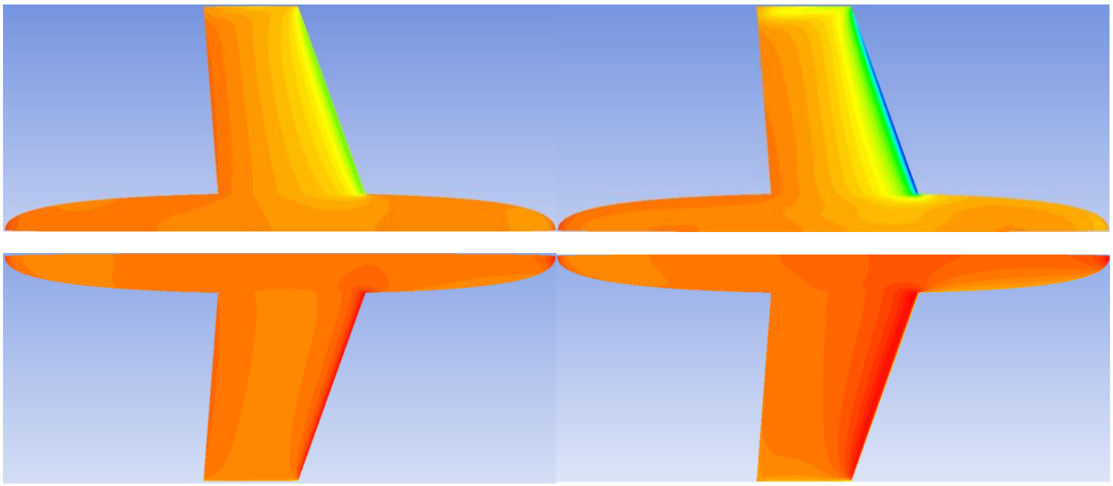
Figure 4 shows the pressure contours on the upper and lower surface of the wing/body obtained with three turbulence models (SA, SST k-omega and WA) at two angles of attack 7° and 16°.



(a) (b)



(c) (d)



(e) (f)

Figure 4: Pressure Contours on the upper and lower surface of wing-body (top part of the figure is the upper surface and the bottom part is the lower surface).
(a) SA model, $\boldsymbol{\alpha} = \mathbf{7}°$ (b) SA model, $\boldsymbol{\alpha} = \mathbf{16}°$ (c) SST k-ω model $\boldsymbol{\alpha} = \mathbf{7}°$ (d) SST k-ω model, $\boldsymbol{\alpha} = \mathbf{16}°$ (e) WA model, $\boldsymbol{\alpha} = \mathbf{7}°$ (f) WA model, $\boldsymbol{\alpha} = \mathbf{16}°$

Table 1 provides the force coefficients obtained with the three turbulence models at two different angles of attack with two different meshes.

Table 1: Force Coefficients

model	alpha	mesh	C_a	C_n	C_z	C_L	C_D	$C_{L,wing}$	$C_{D,wing}$
SA	7	4M	-0.02549	0.4794	0.1127	0.4789	0.03312	0.3934	0.02096
SST	7	4M	-0.02190	0.4900	0.1128	0.4891	0.03799	0.4020	0.02442
SA	7	2M	-0.02514	0.4909	0.1136	0.4904	0.03488	0.4032	0.02145
SST	7	2M	-0.02517	0.4930	0.1135	0.4924	0.03510	0.4052	0.02156
WA	7	2M	-0.02401	0.4909	0.1138	0.4785	0.03481	0.3968	0.01908

SA	16	2M	-0.1538	1.039	0.2536	1.041	0.1384	0.8304	0.08421
SST	16	2M	-0.1797	1.103	0.2686	1.110	0.1313	0.9168	0.08101
WA	16	2M	-0.1673	1.057	0.2619	1.062	0.1305	0.8512	0.07654

Conclusions and Future Work

Numerical simulations are conducted using the CFD simulation software ANSYS FLUENT to determine the aerodynamics characteristics of a wing-body model at two different angles of attack by solving the RANS equations in conjunction with three turbulence models - the SA model, the SST k-ω model, and the WA model. The conclusions based on this study are summarized below.

From Figure 4 and Table 1, the following conclusions can be made:

1) Pressure distributions are almost the same using the SA model, the SST k-ω model and the WA model.
2) Comparing the same aerodynamic coefficient at the same angle of attack, the coefficients obtained with different turbulence models are also nearly the same. The difference in results obtained with the WA model and the other two models is not more than 7.61% which is of similar order of magnitude as the maximum difference of 6.22% between the results from the SA model and the SST k-ω model. This implies that the WA model has reasonable accuracy when simulating the three-dimensional wing-body type flows, when compared to the widely used ST model and SST k-ω model.

In summary, WA model can predict the pressure distribution and force coefficients of flow past a wing-body at subsonic Mach number with the same accuracy as the widely employed SA and SST k-ω turbulence models.

In order to test the general applicability of WA model, more complex geometries such as a wing/body with a multi-element wing and flow conditions such as transonic Mach number and larger angles of attack should be considered.