Prediction of High-Lift Flows Using Two-Dimensional Navier-Stokes Code with Spalart-Allmaras Turbulence Model and Unstructured Multigrids

Yoshiatsu OKI and Akihisa MASUNAGA

1 3rd Research Center, Technical Research and Development Institute, Japan Defense Agency (1-2-10, Sakae-Cho, Tachikawa, 190-8533, Japan)
2 Mitsubishi Heavy Industries, Ltd. (10, Oye-Cho, Minato-Ku, Nagoya, 455-8515, Japan)

A multi-element airfoil flow shows inherently strong complexity for variations in angle of attack and slat and flap settings in high-lift systems. So far, several CFD code validations have been undertaken for high-lift flows. However, in spite of a great deal of efforts, even in two-dimensional computations, the current CFD codes fail to predict flow phenomena that govern high-lift aerodynamics with sufficient accuracy and efficiency. Currently, a two-dimensional compressible Navier-Stokes code with Spalart-Allmaras one-equation turbulence model and an unstructured multigrid method was developed and newly incorporated into a CFD-based design system of TRDI-JDA. For the code validation, the subsonic flows around two- and five-element airfoil configurations are computed. With respect to wing surface pressure coefficient distributions, boundary-layer velocity profiles and forces and moment coefficients, the present computed results are compared quantitatively with the wind-tunnel testing data. As a result, the present code is expected to be a powerful tool to predict and improve high-lift characteristics of multiple elements in aircraft design process.

Key Words: Compressible Flow, CFD, Unstructured Multigrid, Turbulence Model, High-Lift Flow

1. Introduction

A capability for accurate analysis of aircraft high-lift systems performance is an essential requirement for aircraft design process for optimal performance in landing and takeoff phases of flight. Many of the Reynolds-averaged Navier-Stokes (RANS) codes have been already applied for the past decades for the commercial transport aircraft in cruise configuration. However, it has not been established to consistently predict high-lift system performance with sufficient accuracy using computational fluid dynamics (CFD). Development of accurate predictive CFD tools with Navier–Stokes analysis of high-lift flows is indispensable to optimize positioning of leading- and trailing-edge devices and overall system performance in reducing developing cost and expediting the design cycle process.

According to variations in angle of attack and slat and flap positioning in high-lift systems, a multi-element airfoil flow shows highly complex flow phenomena such as laminar flow, attachment line transition, relaminarization, transonic slat flow, confluent boundary-layers, wake interactions, separation, and reattachment (1). A full three-dimensional analysis capability will be finally required for comprehensive computation of high-lift flows with specific three-dimensional phenomena, such as spanwise flow, viscous interference between neighboring aircraft components, etc. However, from a practical point of view, a two-dimensional analysis capability is expected to be useful for predicting high-lift system performance trends on variations of Reynolds number and geometric parameters such as flap and slat settings. A great deal of computational and experimental study on high-lift flows has been performed since 1970s (2). Several CFD code validation efforts have been undertaken ranging form coupled viscous and inviscid methods (3) to incompressible and compressible RANS approaches (4). In a CFD Challenge Workshop held at NASA Langley Research Center in 1993 (5), many CFD results on McDonnell Douglas (MD) three-element configurations have been reported. The compressible Navier-Stokes (N-S) codes, e.g. CFL3D (6), OVERFLOW (4) and TLNS3D (4), as well as the incompressible N-S codes, e.g. INS2D (7) (8), have been extensively used on the structured grid system. Meanwhile, the unstructured flow solver NSU2D (9) has also been applied to the two-dimensional MD three-element configurations. In addition, two-dimensional high-lift flows over three- and four-element configurations were computed using Spalart-Allmaras (S-A) one-equation turbulence model (10) on the unstructured grid system (11).
In this paper, a two-dimensional compressible N-S code with S-A turbulence model and unstructured multigrid method\(^{(12)-(14)}\) was developed and newly incorporated into a CFD-based design system, Computational Aerodynamics System for Performance Evaluation and Research (CASPER)\(^{(15)-(16)}\) of Technical Research and Development Institute of Japan Defense Agency (TRDI-JDA). For the code validation, high-lift flows over two-element configuration, NASA Model-B airfoil, are computed. With regard to wing surface pressure coefficient distributions, boundary-layer velocity profiles and forces and moment coefficients, the present computed results are compared quantitatively with the wind-tunnel testing data\(^{(17)}\). In particular, the advantage of the S-A model is examined in prediction and improvement of high-lift characteristics of over a five-element configuration, NASA Model-D airfoil, the region in which the wake and boundary layer merge. Moreover, performed are the computations of high-lift flows over a five-element configuration, NASA Model-D airfoil, to demonstrate the capability for more complex geometry. Also discussed is the possibility of the present N-S code in prediction and improvement of high-lift characteristics of multiple elements in aircraft design process.

2. Numerical Methods

2.1 Unstructured Grid Generation

An unstructured grid technique has a good flexibility for complex geometry, and so triangular single unstructured grids are applied in the present two-dimensional computations. Semi-structured layer triangular elements are generated using an algebraic method in viscous region near the wing surface and wake. After that, for non-layer inviscid region, triangular unstructured grids are successively generated by the Advancing Front method together with the Delaunay triangulation\(^{(18)}\). As a convergent acceleration procedure, the three-stage V-cycle multigrid procedure\(^{(12)-(14)}\) is applied.

Figures 1 (a) and (b) show the overall and the close-up views of the finest unstructured multigrids of two- and five-element configurations, NASA Model-B and Model-D, respectively. The former contains 69,173 nodes and 137,457 elements. The normalized minimal grid spacing on the wall is a 4.0 \(10^{-5}\) chord, and 11 nodes are in 1.0/Re\(^{1/2}\) (see Fig.1 (b)) with a value of 1.0 for the stretching ratio. To achieve sufficient grid resolution in the boundary layer region, the controlled aspect ratio between the neighboring unstructured grids is approximately 1 to 10 around the wing leading edge for both configurations. Here, the chord, \(C\), is a chord length of wing with multi-element airfoils retracted, and the outer boundary is extended to 40 chords for both configurations.

2.2 Flow Solver

The present governing equations are two-dimensional compressible Reynolds-Averaged Navier-Stokes equations for an ideal gas are written as follows:

\[
\frac{\partial \mathbf{Q}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} + \frac{\partial \mathbf{F}}{\partial y} = \frac{M_a}{Re} \left[ \frac{\partial \mathbf{E}_e}{\partial x} + \frac{\partial \mathbf{F}_e}{\partial y} \right],
\]

where vector of dependent variables, \(\mathbf{Q}\), inviscid flux vectors, \(\mathbf{E}\) and \(\mathbf{F}\), and viscous flux vectors, \(\mathbf{E}_e\) and \(\mathbf{F}_e\), are defined as follows:

\[
\mathbf{Q} = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ e_s \end{pmatrix}, \quad \mathbf{E} = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ (e_s + p)u \end{pmatrix}, \quad \mathbf{F} = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ (e_s + p)v \end{pmatrix},
\]

\[
\mathbf{E}_e = \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{vy} \\ \tau'_{xx} + q_x \end{pmatrix}, \quad \mathbf{F}_e = \begin{pmatrix} 0 \\ \tau_{vy} \\ \tau'_{xy} + q_y \\ \tau'_{xy} + q_y \end{pmatrix}.
\]

In the viscous flux vectors, shear stress tensor \(\tau_{ij}\) and heat flux vector \(q_i\) can be respectively described as follows:

\[
\tau_{ij} = (\mu + \mu_k) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \right),
\]

\[
q_i = \frac{1}{\gamma - 1} \left( \frac{\mu}{Pr} + \frac{\mu_k}{Pr_k} \right) \frac{\partial u^2}{\partial x_i}.
\]

Here, they are nondimensionalized by freestream density, a reference length, and speed of sound \(a\), which is given as follows:

\[
T = a^2 = \gamma (\gamma - 1) \left( \frac{u}{\rho} - \frac{1}{2}(u^2 + v^2) \right).
\]
Fig.1 Overall and close-up views of the finest unstructured multigrids on two- and five-element configurations.

The terms $u$ and $v$ are the Cartesian velocity components in $x$ and $y$ directions, respectively. In addition, the terms $p$, $\rho$, $T$, $k$ and $\varepsilon$ are static pressure, density, temperature, turbulent energy, and total energy per unit volume, respectively. The dimensionless numbers $Pr$, $Pr_t$, $Re$, and $M_\infty$ are Prandtl number assigned a value of 0.72, turbulent Prandtl number assigned a value of 0.90, Reynolds number based on $C$, and freestream Mach number, respectively. The molecular viscosity $\mu$ is determined through the Sutherland’s law as follows:

$$\mu = T^{3/2} \frac{1 + 110.4/T_\infty}{T + 110.4/T_\infty}.$$  

(a) Two-element configuration : NASA Model-B airfoil
(69,173 nodes and 137,457 elements)

(b) Five-element configuration : NASA Model-D airfoil
(256,949 nodes and 511,567 elements)
Moreover, they are closed with the equation of state for an ideal gas as follows:

\[ p = \left( \gamma - 1 \right) \cdot \left[ c_v - \frac{1}{2} \rho \left( u^2 + v^2 \right) \right], \quad (7) \]

where \( \gamma \) is the ratio of specific heats and is prescribed as 1.4 for air.

They are spatially discretized on a triangular unstructured grid system using a cell-centered finite volume approach. The Roe’s flux difference splitting scheme \(^{(19)}\) is employed to evaluate the inviscid flux vectors. As a high resolution scheme, the Frink’s method \(^{(20)}\) is adopted. This is improvement of the conventional Frink’s method \(^{(21)} \) \(^{(22)}\), which interpolated a physical quantity at nodal point to elemental boundary, and then the pseudo-Laplacian weighted averaging interpolation \(^{(23)}\) is implemented in evaluation of each nodal physical quantity with higher accuracy \(^{(24)}\). Meanwhile, the Knight’s method \(^{(25)} \) \(^{(26)}\) using the above pseudo-Laplacian weighted averaging interpolation \(^{(14)}\) is employed to evaluate the viscous flux vectors.

As a time marching algorithm, the LU-SGS (Lower Upper Symmetric Gauss Seidel) implicit scheme \(^{(27)}\) is implemented. Moreover, to accelerate convergence to steady state, the local time stepping and the implicit residual smoothing methods and the three-stage unstructured multigrid approach \(^{(12)} \) \(^{(14)}\) with V-cycle are utilized simultaneously.

### 2.3 Spalart-Allmaras One-Equation Model

The Spalart-Allmaras model (henceforth S-A model) \(^{(10)}\) is a one-equation turbulence model written in terms of the eddy-viscosity-like term \( \tilde{v} \) as follows:

\[
\frac{D \tilde{v}}{Dt} = C_{\mu}(1 - f_{\mu}) \tilde{v} + \frac{M_c}{\sigma \text{Re}} \left[ \nabla \cdot (\tilde{v} + e) \tilde{v} + C_{\mu 2} (\tilde{v}^2) \right] - \frac{M_c}{\text{Re}} \left( C_{\mu 1} f_{\mu} - C_{\mu 1} \tilde{v} \right) \left( \tilde{v}^2 \right) + \frac{\text{Re}}{M_c} f_{\mu} \Delta U^2. \quad (8)
\]

The eddy viscosity \( \mu_e \) is related to the term \( \tilde{v} \) and the kinetic eddy viscosity \( \nu \) through the equation

\[ \mu_e = \rho \nu = \rho \tilde{v} f_{\nu 1}, \quad (9) \]

The production term \( \tilde{S} \) in the above transport equation (8) is given by

\[ \tilde{S} = S + \left( \frac{M_c}{\text{Re}} \right) \left( \frac{\tilde{v}^2}{\kappa^2 d^2} \right) f_{\nu 2}, \quad (12) \]

where \( S \) is the magnitude of the vorticity, and \( d \) is the distance to the wall, and then

\[ f_{\nu 2} = 1 - \frac{\chi}{1 + \chi f_{\nu 1}}, \quad (13) \]

The destruction function \( f_{\nu} \) is given by

\[ f_{\nu} = g \left[ 1 + C_{\nu 1} \frac{1}{d^2} \right], \quad (14) \]

where

\[
\tilde{v} = \tilde{v} \left( \frac{\kappa}{\text{Re}} \right) \left( \frac{\tilde{v}^2}{d} \right) + \frac{\text{Re}}{M_c} f_{\nu} \Delta U^2. \quad (16)
\]

The transition functions are

\[ f_{\nu 1} = C_{\nu 1} g_s \exp \left[ -C_{\nu 2} \left( \frac{\tilde{v}^2}{\Delta U^2} \right) \right], \quad (17) \]

\[ f_{\nu 2} = C_{\nu 3} \exp \left[ -C_{\nu 4} \chi^2 \right], \quad (18) \]

where...
\begin{align}
g(x) = \min \left( 0.1, \frac{\Delta U}{\omega \Delta x} \right).
\end{align}

In the above transition functions, \( d \) is the distance from a field point to the trip on the wall, \( \omega \) is the vorticity at the trip, \( \Delta x \) is the grid spacing at the trip in tangential direction, and \( \Delta U \) is the velocity difference between a field point and the trip. In case of a free transition condition, the trip term of right-hand side of Eq.(8) can be neglected.

The present turbulent parameters are given as follows:

\[
\begin{align*}
C_{b1} &= 0.1355, C_{b2} = 0.622, \\
C_{11} &= 1.0, C_{12} = 2.0, C_{13} = 1.1, C_{14} = 2.0, \\
C_{\omega 1} &= C_{b1} / \kappa^2 + (1 + C_{b2}) / \sigma, C_{\omega 2} = 0.3, \\
C_{\omega 3} &= 2.0, C_{\omega 4} = 7.1, \sigma = 2/3, \kappa = 0.41.
\end{align*}
\]

(20)

3. Results and Discussion

3.1 Two-Element Configuration

Computations on a two-element configuration, NASA Model-B airfoil, were performed at a freestream Mach number of 0.201, angles of attack of 8.00 deg and 8.93 deg, and a Reynolds number based on the chord length of wing with a flap retracted of 2.83 \( \times 10^6 \). Here, total number of the time iterations for convergence was 25,000 with a CFL number of 5.0, which costs about 10.0 CPU hours on a single processor of the NEC SX-4/2C supercomputer.

Figure 2 shows the Mach number contours at a Mach number of 0.201, an angle of attack of 8.93 deg, and a Reynolds number of 2.83 \( \times 10^6 \). Boundary layers on main-element upper surface, the flow acceleration at gap between main-element and flap and confluent main-element wakes and flap boundary layers ranging from a main-element trailing edge to flap upper surface can be well simulated. As is evident from \( \text{Cp}^* \) in Fig.4 mentioned below, a slight transonic flow can be locally observed at a main-element leading edge. In most of the unstructured computations, a contour line becomes ragged due to triangular unstructured grids. However, the present unstructured computed contour line is obtained smoothly, which suggests that the high-resolution procedure works well.

Figure 3 shows the eddy viscosity contours at a Mach number of 0.201, an angle of attack of 8.93 deg, and a Reynolds number of 2.83 \( \times 10^6 \). Here, \( \text{Cp}^* \) is a freestream molecular viscosity. We can confirm turbulent boundary layers developed along the main-element upper surface, difference of boundary layer thickness between upper and lower surface and computational domain influenced by the turbulence model. Moreover, inviscid flows are dominant in region between main-element wake and flap upper surface, and then this optimization is essential in high-lift aerodynamic design.

Figure 4 shows the comparisons of computed and experimental wing pressure coefficient distributions at a Mach number of 0.201, an angle of attack of 8.93 deg, and a Reynolds number of 2.83 \( \times 10^6 \). Here, \( \text{Cp}^* \) is a critical pressure coefficient of -16.3. The present N-S computed results agree better with the wind-tunnel testing data (17) than
the Euler computed results in both the main-element and flap, and then a main-element leading edge suction peak is also well captured.

Figures 5 shows the comparisons of computed and experimental boundary layer velocity profiles for five survey stations of STN6 to STN10 at a freestream Mach number of 0.201, an angle of attack of 8.00 deg, and a Reynolds number of 2.83 $\times 10^6$, respectively. Here, the horizontal axis, $U/U_\infty$, is a dimensionless boundary layer velocity, and the vertical axis, $Y_n/C$, is a dimensionless height from wing surface. In STN6 of a main-element upper surface, the present computed turbulent boundary layer is in good agreement with the wind-tunnel testing data $^{(17)}$. Turning to STN7 to STN10 of flap upper surface, the computed center position of main-element wake shows difference of about 0.3% chord in STN7 right after a spout between main-element and flap, as compared to the wind-tunnel testing data, and the computed boundary layers on flap upper surface are thickly overestimated because . On the whole, however, confluent wakes and boundary layers are well simulated.

Fig.4 Comparison of computed and experimental wing surface pressure coefficient distributions for NASA Model-B airfoil at $M_\infty=0.201$, $\alpha=8.93$ deg and $Re=2.83\times10^6$.

Fig.5 Comparison of computed and experimental boundary layer velocity profiles for NASA Model-B airfoil at $M_\infty=0.201$, $\alpha=8.00$ deg and $Re=2.83\times10^6$. 

- 6 -
As is evident from the experimentally obtained velocity profiles on flap upper surface, a flap upper surface boundary layer turns to laminar owing to the spout around flap leading edge in STN7 and STN8 and shifts to turbulent boundary layer halfway on the flap upper surface in STN9 and STN10.

As shown in Fig.3, there exists no eddy viscosity in inviscid potential core of flap upper surface. On the other hand, the turbulent boundary layer is fully developed in the main-element wake and so the eddy viscosity is larger than that of the boundary layer on flap upper surface.

Table 1 shows the comparison of computed and experimental forces and moment coefficients at a freestream Mach number of 0.201, an angle of attack of 8.93 deg, and a Reynolds number of \(2.83 \times 10^6\). Here, parenthetic values are difference between the computed results and the wind-tunnel testing data\(^{(17)}\). Thus, the N-S computed drag coefficients are slightly different from the wind-tunnel testing data, and the N-S computed lift and moment coefficients are better agreement with the wind-tunnel testing data.

Table 1  Comparison of experimental and computed forces and moment coefficients for NASA Model-B airfoil at \(M_\infty=0.201\), \(\alpha=8.93\) deg and \(Re=2.83\times10^6\).

<table>
<thead>
<tr>
<th></th>
<th>Lift coefficient (C_l)</th>
<th>Drag coefficient (C_d)</th>
<th>Moment coefficient (C_m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>WTT</td>
<td>2.580 ( - )</td>
<td>0.0400 ( - )</td>
<td>-0.400 ( - )</td>
</tr>
<tr>
<td>Euler Comp.</td>
<td>2.733 ( 0.153 )</td>
<td>0.0894 ( 0.0494 )</td>
<td>-0.444 ( 0.044 )</td>
</tr>
<tr>
<td>N-S Comp. (S-A)</td>
<td>2.660 ( 0.080 )</td>
<td>0.0538 ( 0.0138 )</td>
<td>-0.411 ( 0.011 )</td>
</tr>
</tbody>
</table>

3.2 Five-Element Configuration

To confirm the possibility of the present code for more complex geometries, computations on a five-element configuration, NASA Model-D airfoil, were also performed at a freestream Mach number of 0.201, angles of attack of 0.01 deg and 8.17 deg, and a Reynolds number based on the chord length of wing with a slat and three flaps retracted of \(2.83 \times 10^6\). Here, total number of the time iterations for the convergence was 60,000 with a CFL number of 3.0, which costs about 99 CPU hours on a single processor of the NEC SX-4/2C supercomputer.

Figure 6 shows the convergence history at a freestream Mach number of 0.201, an angle of attack of 8.17 deg, and a Reynolds number of \(2.83 \times 10^6\). Because of hard computational condition of convergence such as the subsonic and five-element airfoil flow, 60,000 multigrid-cycles are required for convergence. Nevertheless, convergence characteristic is good, and the present code is robust. Here, convergent criteria for forces and moment coefficients are within \(C_l \leq 0.0005\), \(C_d \leq 0.00005\), and \(C_m \leq 0.0005\), respectively. In particular, the convergent criterion of drag coefficient is severely set.

Figure 7 shows the Mach number contours at a Mach number of 0.201, an angle of attack of 8.17 deg, and a Reynolds number of \(2.83 \times 10^6\). A leading edge slat wake effects the whole region on main-element upper surface. Besides, a forward flap wake reaches down to the backward flap as well as the middle flap.

Figure 8 shows the eddy viscosity contours at a Mach number of 0.201, an angle of attack of 8.17 deg, and a Reynolds number of \(2.83 \times 10^6\). An intensive eddy viscosity domain can be observed around the concave of rear main-element lower surface and backward flap downstream. In the same way as the two-element configuration, the inviscid flows are dominant through slat wake and main-element upper surface, forward flap wake and middle flap upper surface, and middle flap wake and backward flap upper surface.

Fig.6 Convergence history for NASA Model-D airfoil at \(M_\infty=0.201\), \(\alpha=8.17\) deg and \(Re=2.83\times10^6\).
Fig. 7  Computed Mach number contours for NASA Model-D airfoil at $M_\infty=0.201$, $\alpha=8.17$ deg and $Re=2.83\times10^6$.

Fig. 8  Computed eddy viscosity contours for NASA Model-D airfoil at $M_\infty=0.201$, $\alpha=8.17$ deg and $Re=2.83\times10^6$.

Figures 9 (a) and (b) show the comparisons of computed and experimental wing pressure coefficient distributions at a Mach number of 0.201, angles of attack of 0.01 deg and 8.17 deg, and a Reynolds number of $2.83\times10^6$, respectively. The difference between the present N-S results and the wind-tunnel testing data (17) on the wing upper surface pressure coefficient distributions of each element is approximately 0.2 to 0.3 with the exception of the backward flap in both angles of attack. However, the complicated surface pressure coefficient distributions of each element are well simulated by the present code.

4. Conclusions

To implement high-lift aerodynamic design for advanced aircraft at the next generation with sufficient computational accuracy and efficiency, a two-dimensional compressible unstructured N-S code with S-A model was incorporated into the CASPER. For the code validation, the computations of subsonic flows around two- and five-element configurations were performed. The present computed results were quantitatively compared with the other
computed results and the wind-tunnel testing data. These results lead us to the following conclusions:

1. From the Mach number and eddy viscosity contours, confluent boundary-layers and wakes can be well simulated with good efficiency in high-lift system of multiple elements, slat, main element and flap by using the present code with S-A model.

2. As is evident from the comparisons of wing surface pressure distributions, the present N-S computed results with S-A model agree better with the wind-tunnel testing data.

3. In the longitudinal forces and moment prediction, the present N-S computed drag coefficient was improved.

4. In comparison of turbulence models on boundary-layer velocity profiles of NASA model-B airfoil, S-A model underestimates a slight velocity gradient on wing surface but agrees well with the wind tunnel testing data in the region in which the wake and boundary-layer merge.

References


(22) Frink, N. T., Parikh, P., and Pirzadeh, S., “A Fast Upwind Solver for the Euler Equations on


