ABSTRACT

This paper presents detailed description of a novel CFD procedure and comparison of its solution results to that obtained by other available CFD codes as well as actual flight and wind tunnel test data pertaining to the GIII aircraft, currently undergoing flight testing at AFRC.

INTRODUCTION

Two in-house\(^1\) software as well as a number of commercially\(^2-6\) available CFD codes were used to analyze the problem, for comparison purposes. In this process both finite volume and finite element discretization were used for Euler and Navier-Stokes simulations. Both unstructured and structured grids were employed, as appropriate and solutions were derived for Mach 0.701 and angle of attack \(\alpha = 3.92\) degree.

Extensive flight tests were performed for validation purposes. Also these tests were complimented with detailed wind tunnel simulations. All such test results are compared with the numerical solution data obtained by the various CFD codes. Associated finite difference\(^7\) and finite volume\(^8,9\) techniques are well described in the literature\(^10,11\). The finite element technique\(^12\) for the discretization of fluid flow employs unstructured grid and is based on a Taylor-Galerkin procedure\(^13-15\).

A description of the finite element fluids solver is presented in some detail. It pertains to the solution of viscous flow represented by the Navier-Stokes formulation. An unstructured grid is used for domain decomposition.

The one equation model (Ref. 16) has been adapted for turbulence modelling. In this process both the viscous stresses pertaining to the linear viscous flow and the flux in the energy equation, are duly modified.

It is then followed by detailed results of analyses which are next compared with actual flight test and wind tunnel simulation results. These results indicate that most CFD solutions compare reasonably well with the test data. The FE solutions in particular prove to be efficient and accurate and the related software are available for public use.

Finally, some summarizations and discussions of the current effort is given in the ‘Concluding Remarks’ section.

NOMENCLATURE

\begin{align*}
\text{AFRC} &= \text{Armstrong Flight Research Center} \\
\text{CFD} &= \text{Computational Fluid Dynamics} \\
\text{FE} &= \text{Finite Element} \\
\Delta t &= \text{time step} \\
\rho &= \text{Density} \\
\mu &= \text{Dynamic viscosity} \\
\sigma &= \text{Viscous stress tensor} \\
u &= \text{free stream velocity} \\
E &= \text{Total energy} \\
\mathbf{a} &= \text{Shape function} \\
\mathbf{v} &= \text{Conservation variable} \\
f &= \text{Convection} \\
g &= \text{Diffusion}
\end{align*}
\[ k = \text{Thermal conductivity} \]
\[ p = \text{Pressure} \]
\[ M = \text{Mass matrix} \]
\[ K = \text{Convection matrix} \]
\[ Re = \text{Reynolds number} \]
\[ Pr = \text{Prandtl number} \]

**PROCEDURE**

The Navier-Stokes equation can be written as
\[
\frac{\partial v}{\partial t} + \frac{\partial f_j}{\partial x_j} + g_j = 0 \quad i = 1, 2, 3
\]  
(1)
in which the conservation variables, flux, and body force column vectors, as well as the viscous stress are defined as
\[
v = [p \quad \rho u_j \quad \rho E]^T, \ j = 1, 2, 3
\]  
(2)
\[
f_j = [\rho u_j] \quad (\rho u_i u_j + \delta_{ij}) \quad u_j (p + \rho E)]^T, \ j = 1, 2, 3
\]  
(3)
\[
E_j = \begin{bmatrix} 0 \quad \sigma_{ij} \quad (u_i \sigma_{ij} + k \frac{\partial T}{\partial x_i}) \end{bmatrix}^T
\]  
(4)
\[
f_b = \begin{bmatrix} 0 \quad f_{b_i} \quad u_j f_{b_j} \end{bmatrix}
\]  
(5)
\[
\sigma_{ij} = \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \frac{2}{3} \frac{\partial u_l}{\partial x_i} \delta_{ij}
\]  
(6)
where \(u_i\) are velocity components in the \(x_i\) coordinate system; \(\rho, p, E\) are the density, pressure, and total energy respectively; \(\mu\) is the dynamic viscosity; \(k\) is the thermal conductivity; \(q_j\) is the heat flux being \(-k \partial T/\partial x_j\); \(T\) is the temperature; \(f_j\) represents the body forces.

The preceding equations are nondimensionalised for numerical calculations. In this process the governing equations remain in the same form excepting \(g_j\), which becomes
\[
g_j = \begin{bmatrix} 0 \quad \sigma_{ij} \quad (u_i \sigma_{ij} - q_j) \end{bmatrix}^T
\]  
(7)
and also the viscous stress tensor and heat flux take the following form:
\[
\sigma_{ij} = \mu \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] \frac{2}{3} \frac{\partial u_l}{\partial x_i} \delta_{ij}
\]  
(8)
in which the Reynolds number is defined as \(Re = u \omega L/\nu_\omega\); \(\nu_\omega = \mu_\omega/\rho_\omega\) is termed the kinematic viscosity; \(Pr\) is the Prandtl number; \(Pr = \nu_\omega/\alpha_\omega\), with \(\alpha_\omega = k/(\rho_\omega c_p)\) is the thermal diffusivity.

The Taylor’s expansion of the solution \(v(x, t)\) in the time domain, neglecting second order term and body forces, yields
\[
\Delta v = -\Delta t \begin{bmatrix} f_{1i} \quad \partial q_i \end{bmatrix}^T
\]  
(9)
in which \(\Delta v = v(t + \Delta t) - v(t)\). Applying Galerkin’s spatial idealization \(v = \tilde{v} \Phi, \tilde{v}\) being the nodal values and \(\Phi\) the shape functions vector, the flow equation can be expressed as
\[
M \Delta \tilde{v} = -\Delta t \begin{bmatrix} f_{1i} \\ M + K \end{bmatrix} \tilde{v} - \Delta t (\tilde{f}_1 + \tilde{f}_2) + \Delta t \tilde{R} + \Delta t [K_\sigma + f_\sigma]
\]  
(10)
in which \(M\) is the consistent mass matrix, \(K\) the convection matrix, \(\tilde{f}_1, \tilde{f}_2\) the pressure matrices, \(K_\sigma\) the second-order matrix that includes viscous and heat flux effects, and \(f_\sigma\) the boundary integral matrix from second-order terms. Then,
\[
M = \int_v a^T \alpha dV; \quad K = \int_v a^T u_i \frac{\partial a}{\partial x_i} dV;
\]
\[
\tilde{f}_1 = \int_v a^T \tilde{f}_1 \frac{\partial e_i}{\partial x_i} dV; \quad \tilde{f}_2 = \int_v a^T \tilde{e}_i \frac{\partial e_i}{\partial x_i} dV;
\]
\[
K_\sigma = -\int_v \frac{\partial a^T}{\partial x_j} e_j \sigma_{ij} dV - \int_v \frac{\partial a^T}{\partial x_j} m_j q_j dV;
\]
\[
f_\sigma = \int_v \tilde{a}^T \epsilon_{ij} \sigma_{ij} \tilde{n} d\Gamma + \int_v \tilde{a}^T m_j q_j \tilde{n} d\Gamma
\]  
(11)
In these equations, \(\tilde{p}_i, \tilde{u}_i, \tilde{e}_i\) are the average values; \(e_i = [0 \ 1 \ 0 \ 0 \ u_i]^T, e_2 = [0 \ 0 \ 1 \ 0 \ u_2]^T, e_3 = [0 \ 0 \ 0 \ 1 \ u_3]^T\), \(R\) is the artificial dissipation, and \(m_1 = m_2 = m_3 = [0 \ 0 \ 0 \ 0 \ 1]^T\). Turbulence terms are included by modifying the viscous effects.

A novel two-step solution procedure is adopted for the flow equation, the inviscid solution being augmented with the viscous term and stabilized with artificial dissipation terms. Assuming,
\[
\Delta \tilde{v} = \tilde{v}_{n+1} - \tilde{v}_n
\]  
(12)
then,
\[
M(\tilde{v}_{n+1} - \tilde{v}_n) = \frac{\Delta t}{2} [cM + K](\tilde{v}_{n+1} + \tilde{v}_n) - \Delta t(\tilde{f}_1 + \tilde{f}_2)
\]  
(13)
which becomes
\[
\left(1 + \frac{\Delta t}{2} c\right)M + \frac{\Delta t}{2} K \tilde{v}_{n+1} = \left(1 - \frac{\Delta t}{2} c\right)M - \frac{\Delta t}{2} K \tilde{v}_n + \Delta t R
\]  
(14)
or
\[
[M+] \tilde{v}_{n+1} = [M-] \tilde{v}_n + \Delta t R
\]  
(15)
where
\[
R = -\left(\tilde{f}_1 + \tilde{f}_2\right)
\]  
(16)
Let
\[
M_+ = D_+ + M_+' \quad \text{and} \quad \text{the matrix } D_+ \text{ having diagonal elements. Equation (8) may then be solved as follows.}
\]
Step 1: Form
\[
[D_+] \tilde{v}_{n+1} = [M-] \tilde{v}_n - [M+] \tilde{v}_{n+1} + \Delta t R
\]  
(17)
Step 2: Solve \(\tilde{v}_{n+1}\) iteratively
\[
\tilde{v}^{(i+1)} = [D_+]^{-1} \left( [M-] \tilde{v}_n - [M+] \tilde{v}^{(i)}_{n+1} + \Delta t (R + \tilde{R} + K_\sigma + f_\sigma) \right)
\]  
(19)
Step 3: If \(\|\tilde{v}^{(i+1)}\| \neq \text{EPS1}\|\tilde{v}^{(i)}_{n+1}\|\) go to Step 2.
Step 4: If \(\|\tilde{v}^{(i+1)}\| \neq \text{EPS2}\|\tilde{v}^{(i)}_{n+1}\|\) go to Step 1.
Step 5: Repeat Steps 1 to 4 NITER times until desired convergence is achieved, that is until \(\tilde{v}_{n+1} \approx \tilde{v}_n\); EPS1 and EPS2 are suitable convergence criteria factors, specified by the users.

The iterative process in Step 2 requires a small number of steps, usually 1, and achieves a stable, convergent solution. In regions of high pressure gradients, artificial dissipation term is applied to prevent oscillations near discontinuities. This is implemented by incorporating pressure-switched diffusion coefficients as appropriate. Thus,
\[
\tilde{R} = \frac{c_s S_p}{\Delta t} M^{-1}_c [M_c - M_p] \tilde{v}_n
\]  
(20)
in which \(c_s\) is a shock capturing constant, \(S_p\) is the averaged element value of the nodal pressure switch defined as
\[ S_i = \frac{\left| \sum p_i - p_j \right|}{\sum \left| p_i - p_j \right|} \]  \quad (21)

and \( M_c \) and \( M_l \) are the consistent and lumped mass matrices respectively; \( i \) is the node under consideration and \( j \) are the nodes connected to \( i \).

To obtain the viscous components, \( \sigma_{ij} \) in Eq. (4) is written as

\[ \sigma_{ij} = -\frac{2}{\text{Re}} \frac{\partial u_i}{\partial x_j} \delta_{ij} + \frac{\mu}{\text{Re}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  \quad (22)

and the diffusion flux of the Navier-Stokes equation being

\[ g_i = \begin{pmatrix} \sigma_{i1} & \sigma_{i2} & \sigma_{i3} & u_i \sigma_{ij} + \frac{1}{\text{RePr}} \frac{\partial T}{\partial x_j} \end{pmatrix}^T \]

\( i = 1,2,3; \quad j = 1,2,3 \)  \quad (23)

\( \mu \) is the nondimensional viscosity term, whereas \( \text{Re} \) and \( \text{Pr} \) are the Reynolds and Prandtl numbers, respectively. Next components of \( \partial g_i/\partial x_j \) are evaluated term by term and then discretized by Galerkin approximation.

This procedure is adopted in the STARS-CFD code\(^1\) that enables effective solution of the Navier-Stokes equation in most flight regimes.

**NUMERICAL AND TEST RESULTS**

Accuracy\(^{18}\) of the STARS CFD code was verified pertaining to the Hyper-X flight vehicle, carrying the X-43 vehicle for subsequent hypersonic flight at Mach 5.0 and 7.0. Table 1 provides such a comparison of computational results and actual flight test data at various sensor locations; these data pertain to the ascent state of Hyper-X at Mach 0.9 and an altitude of 22,500 ft. Figure 1 provides a graphical depiction of comparison of the two sets of results, signifying accuracy of the relevant procedures. Also Table 1 shows the numerical values of flight test and computed aerodynamic pressures; excellent correlation is observed for primary data values; the last three values in the Table are comparatively small and hence prone to measurement inaccuracy. This code was next used, along with a variety of existing commercially available programs, to solve a practical project problem. The results of which were also compared to that obtained by actual flight and wind tunnel tests.

The Gulfstream GIII airplane (Gulfstream Aerospace Corporation, Savannah, Georgia), currently undergoing flight tests\(^{19}\) at NASA AFRC, was chosen as the example problem for verification purposes. The GIII business jet as shown in Figure 2 is being modified and instrumented by NASA’s Armstrong Flight Research Center to serve as a test bed for a variety of flight research experiments, in support of the Environmentally Responsible Aviation (ERA) project. The twin-turboprop aircraft provides long-term capability for efficient testing of subsonic flight experiments for NASA, the U.S. Air Force, other government agencies, academia, and private industry.

The wing span of the GIII aircraft is 23.7226 meter with sweep angle 27.66 degree. The airfoil section is a NACA 0012 modification. The aerodynamic model of the GIII wing used in the CFDSOL and MG solutions is shown in Figure 3; only the right wing section was used for CFD analysis. Total number of CFD mesh using triangular element on wing surface is 31k for coarse mesh and 59k for finer mesh. Total number of 3-D CFD mesh using tetrahedron element in aerodynamic domain is 1.2m for coarse mesh and 2.8m for finer mesh.

The flight condition was for Mach 0.7 and angle of attack \( \alpha = 3.92 \) degrees. Table 2 provides extensive description of relevant analyses hardware employed for each of the participative code and solution CPU time for a converged solution. The STARS has two solution option modules, namely CFDSOL and MG and both appear to be competitive in terms of solution time, accuracy, grid size and CPU numbers.

Figures 4 to 6 depict pressure (\( C_p \)) distribution around the wing airfoil cross section at the wing 368.3 cm, 584.2 cm and 1003.5 cm span wise locations. Further, the wind tunnel and actual flight test results are also shown for comparison and validation purpose. Due to the proprietary nature of the wind tunnel and flight test data, actual scales on the figures cannot be shown. Each of the codes shows reasonable correlation; solution of the CFDSOL and MG codes appear to be rather close to the two test results.

Figure 7 depicts the \( C_p \) distribution along the airfoil at different span locations.

**CONCLUDING REMARKS**

The paper presents detailed comparison of solutions of the GIII aircraft wing obtained by a number of commercially available CFD codes as well as two AFRC in-house codes that use a finite element fluids discretization employing unstructured grids; related formulations of the novel CFDSOL code are also presented in detail. Importantly these solutions are compared with actual flight and also wind tunnel test data. Each of the codes shows reasonable correlation; solution of the CFDSOL and MG codes appear to be rather close to the two test results, particularly around the leading edge; further, use of a single CPU to derive solutions testifies to their cost effectiveness.

**REFERENCES**


6. CD-adapco, STAR-CCM +


Table 1 Comparison of computed and flight test measured pressure data for the Hyper-X/X-43 vehicle

<table>
<thead>
<tr>
<th>Sensor point</th>
<th>Flight test</th>
<th>CFD computed</th>
<th>Percent difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>001</td>
<td>0.01165</td>
<td>0.01193</td>
<td>2.34</td>
</tr>
<tr>
<td>003</td>
<td>0.01227</td>
<td>0.01164</td>
<td>6.12</td>
</tr>
<tr>
<td>007</td>
<td>-0.00167</td>
<td>-0.00096</td>
<td>42.12</td>
</tr>
<tr>
<td>085</td>
<td>-0.00108</td>
<td>-0.00268</td>
<td>147.99</td>
</tr>
<tr>
<td>090</td>
<td>0.00048</td>
<td>-0.00055</td>
<td>2.56</td>
</tr>
</tbody>
</table>

Table 2 CFD Solvers Comparison

<table>
<thead>
<tr>
<th>CFD Solver</th>
<th>Flow Equation</th>
<th>Platform</th>
<th>No. of CPU</th>
<th>Total CPU time</th>
<th>Grid Size</th>
<th>Note</th>
</tr>
</thead>
<tbody>
<tr>
<td>STARCCM+</td>
<td>RANS, finite volume, K-omega SST turbulence</td>
<td>Cluster</td>
<td>~80</td>
<td>6hr, 40min (533 cpu hours) - 3000 iterations</td>
<td>7.2M polyhedra/prismatic for half model without T-tail</td>
<td>number of processors is an estimate, and the time is an estimate for that number of processors</td>
</tr>
<tr>
<td>STARS (MG)</td>
<td>Euler, finite element</td>
<td>Dell M620 8GB Ram, 64 bit</td>
<td>1 Intel Core i7 @2.67 GHz</td>
<td>2.8 hr, (100 steps, 25 inner cycles)</td>
<td>1.2 M Tetrahedrons for wing only</td>
<td></td>
</tr>
<tr>
<td>STARS (CFDSOL)</td>
<td>Full N-S, finite element</td>
<td>Dell M620 8GB Ram, 64 bit</td>
<td>1 Intel Core i7 @2.67 GHz</td>
<td>13.8 hr (10000 steps)</td>
<td>2.8 M Tetrahedrons for wing only</td>
<td></td>
</tr>
<tr>
<td>USM3D</td>
<td>Full N-S, finite volume</td>
<td>Mac 64 bit</td>
<td>2 CPUs</td>
<td>16 hr</td>
<td>1.9 M cells for half model without T-tail</td>
<td></td>
</tr>
<tr>
<td>TRANAIR</td>
<td>Full potential + viscosity (boundary layer)</td>
<td>Linux Workstation</td>
<td>1 CPU</td>
<td>2h, 28min</td>
<td>1.7M cells for full model</td>
<td></td>
</tr>
</tbody>
</table>
Fig. 1 Comparison of flight measured and calculated (CFD) pressure on Hyper-X/X-43 vehicle

Fig. 2 Grumman Gulfstream III (GIII) business jet.
Fig. 3 Aerodynamic model of the GIII aircraft wing.
Fig. 4 $C_p$ plot at span station 145

Fig. 5 $C_p$ plot at span station 230
Fig. 6 $C_p$ plot at span station 385.

(a) $C_p$ distribution on wing surface

(b) $C_p$ at station 145

(c) $C_p$ at station 230

(d) $C_p$ at station 385

Fig. 7 Typical $C_p$ plots at various locations