Progress on hybrid unsteady simulation of helicopter rotor flow

By S. Hahn, J. Alonso, J. Baeder†, K. Duraisamy†, G. Iaccarino, S. Lele, P. Moin, F. Schmitz†, M. Shoeybi and X. Wu

1. Motivation and objectives

Central to the computation of helicopter blade noise is the accurate prediction of blade-wake interaction (BWI) and blade-vortex interaction (BVI) surrounding the rotor. Numerical discretization errors in the wake region have to be minimized in order to capture the vortex evolution with high fidelity. In particular, artificial dissipation introduced in the numerical schemes has a direct influence on the dissipation of the vortex cores and its subsequent spreading. Unfortunately, most existing rotorcraft RANS solvers are based on formulations that possess considerable amounts of numerical dissipation added for various purposes such as shock capturing. Recently, a low-dissipation algorithm has been developed for LES at Stanford University by Mahesh et al. (2004) and Ham & Iaccarino (2004). The name of the associated LES flow solver is CDP. In principle, this low-dissipation algorithm can be extended to a RANS context and may provide a better alternative for resolving the flow in the wake. CDP has been successfully coupled with a fully compressible multiblock-structured RANS solver SUmb for jet engine flow simulation. Numerical algorithm used in the structured RANS solver SUmb is described in Alonso et al. (2002). The objective of this work is to develop and validate a robust procedure of integrating SUmb for flow in the neighborhood of the blade and a low-dissipation, LES-like, CDP-based unstructured RANS solver in the wake region. This combination provides a more promising approach than traditional rotorcraft flow solvers by resolving both compressibility and wake effects with solvers best suited for each purpose. To this end, considerable amounts of code development and validation are necessary, and some of these efforts are described in this article.

2. Definition of grid motion

Computation of flow over helicopter rotor blades with generalized mode of operation, e.g. forward flight, requires moving mesh capability in the solver. Let \( V(t) \) be a moving control volume with bounding surface \( S(t) \) and outward unit normal vector \( \mathbf{n} \). \( \mathbf{r} \) is the position vector of a point in space with respect to an inertial reference frame. The local boundary velocity is \( \mathbf{b} \). It is often convenient to define the boundary geometry with respect to a non-inertial reference frame. Let \( \mathbf{r}_0(t) \), \( \mathbf{b}_0(t) \), and \( \mathbf{\Omega}(t) \) be the position vector of the origin, velocity, and angular velocity of the non-inertial frame relative to the fixed inertial frame. Then the boundary velocity can be written as

\[
\mathbf{b} = \mathbf{b}_{de} + \mathbf{b}_{tr},
\]

† Alfred Gessow Rotorcraft Center, University of Maryland
where the velocity due to translation and rotation is

$$\mathbf{b}_{tr} = \mathbf{b}_0 + \mathbf{\Omega}(t) \times [\mathbf{r} - \mathbf{r}_0(t)] .$$

(2.2)

3. RANS governing equations of CDP with moving mesh

Let $\mathbf{u}$ be the ensemble-averaged velocity fluid velocity at $\mathbf{r}$ in a fixed inertial reference frame. The continuity equation for an incompressible fluid in the moving control volume $V(t)$ is

$$\frac{d}{dt} \int_{V(t)} \rho dV + \int_{S(t)} \rho (\mathbf{u} - \mathbf{b}) \cdot \mathbf{n} dA = 0 .$$

(3.1)

For CDP only we apply the assumption of non-deforming grids with solid-body rotation and translation. The surface velocity relative to the non-inertial reference frame is

$$\mathbf{b}_{def} = 0 .$$

(3.2)

Since the volume change of an element with solid-body rotation and translation is zero, as in Vinokur (1989), we have the important relation

$$\int_{S(t)} \mathbf{b} \cdot \mathbf{n} dA = 0, \quad \nabla \cdot \mathbf{b} = 0 .$$

(3.3)

For the incompressible fluid considered in this work, and with the assumption that volume change of the moving cell is zero,

$$\frac{d}{dt} \int_{V(t)} \rho \mathbf{u} dV = 0 ,$$

(3.4)

so that

$$\int_{S(t)} (\mathbf{u} - \mathbf{b}) \cdot \mathbf{n} dA = 0 .$$

(3.5)

Of course for incompressible flow regardless of mesh motion at any instant we always have

$$\int_{S(t)} \mathbf{u} \cdot \mathbf{n} dA = 0 .$$

(3.6)

Unsteady incompressible RANS momentum equations for fluid in the moving cell are

$$\frac{d}{dt} \int_{V(t)} \rho \mathbf{u} dV + \int_{S(t)} \rho \mathbf{u} (\mathbf{u} - \mathbf{b}) \cdot \mathbf{n} dA = - \int_{S(t)} p n dA + \int_{S(t)} \mathbf{T} dA$$

(3.7)

$$\mathbf{T} = \left\{ (\mu + \mu_t) \left[ \text{grad} \mathbf{u} + (\text{grad} \mathbf{u})^T \right] \right\},$$

(3.8)

where $T$ denotes transpose operation. $\mu$ and $\mu_t$ is molecular and turbulent dynamic eddy viscosity, respectively.

4. RANS governing equations of SUmb with moving mesh

The fully compressible multiblock-structured RANS solver SUmb allows more general types of grid motion, including deformation. Continuity and unsteady Reynolds-averaged
governing equations associated SUnm are
\[ \frac{d}{dt} \int_{V(t)} \rho dV + \int_{S(t)} \rho (\mathbf{u} - \mathbf{b}) \cdot \mathbf{n} dA = 0 \] (4.1)
\[ \frac{d}{dt} \int_{V(t)} \rho \mathbf{u} dV + \int_{S(t)} \rho \mathbf{u} (\mathbf{u} - \mathbf{b}) \cdot \mathbf{n} dA = -\int_{S(t)} \rho \mathbf{n} dA + \int_{S(t)} T dA + \int_{S(t)} \sigma dA , \] (4.2)
where \( T = \left\{ (\mu + \mu_t) \left[ \text{grad} \mathbf{u} + (\text{grad} \mathbf{u})^T \right] \right\} \), and \( \sigma = -\frac{2}{3} \left\{ (\mu + \mu_t) \text{div} \mathbf{u} \right\} \).

5. Moving mesh and kinetic energy conservation in CDP

It is of interest to investigate whether the discrete kinetic energy conservation property in unstructured, fractional step scheme developed by Mahesh et al. (2004) for stationary grids remains valid for non-deforming grids with solid-body rotation and translation. Four main arguments were made in Mahesh et al. with reference to the derivation of the property of kinetic energy conservation in their scheme. These are:

(a) When properly formulated, discrete conservation of a passive scalar in incompressible flows leads to the discrete conservation of the square of the passive scalar.

(b) Fractional step method can be formulated with combined usages of cell center based Cartesian velocity and face center based face normal velocity.

(c) For non-staggered grids, although pressure gradient term is non-conservative in the equation of discrete kinetic energy, its effect can be minimized through proper evaluation of the pressure gradient at cell center.

(d) For staggered grids, pressure gradient term is conservative in the equation of discrete kinetic energy.

In this section we will demonstrate that the first three points still hold true in non-deforming grids with solid-body rotation and translation. The last point is not relevant to this work, because we are using the collocated formulation of CDP.

For the problem we are considering, transport equation for a passive scalar is
\[ \frac{d}{dt} \int_{V(t)} \phi dV + \int_{S(t)} \phi (\mathbf{u} - \mathbf{b}) \cdot \mathbf{n} dA = 0 . \] (5.1)
According to Vinokur (1988), the differential formulation of (5.1) is
\[ \frac{1}{V(t)} \frac{\partial [V(t) \phi]}{\partial t} + \nabla \cdot [\phi (\mathbf{u} - \mathbf{b})] = 0 . \] (5.2)
Writing out the left-hand-side of (5.2) for variable \( \phi^2 \), we have,
\[ \frac{1}{V(t)} \frac{\partial [V(t) \phi^2]}{\partial t} + \nabla \cdot [\phi^2 (\mathbf{u} - \mathbf{b})] = 2\phi \frac{\partial \phi}{\partial t} + \phi \frac{\phi}{V(t)} \frac{\partial V}{\partial t} + \phi^2 \nabla \cdot (\mathbf{u} - \mathbf{b}) + (\mathbf{u} - \mathbf{b}) \cdot \nabla \phi^2 \]
\[ = 2\phi \left\{ \frac{1}{V(t)} \frac{\partial [V(t) \phi]}{\partial t} + \nabla \cdot [\phi (\mathbf{u} - \mathbf{b})] \right\} \]
\[ = 0 . \] (5.3)
The integral equivalent of (5.3) is
\[ \frac{d}{dt} \int_{V(t)} \phi^2 dV + \int_{S(t)} \phi^2 (\mathbf{u} - \mathbf{b}) \cdot \mathbf{n} \, dA = 0. \quad (5.4) \]
Thus continuous conservation of \( \phi \) also implies continuous conservation of \( \phi^2 \) in the present moving mesh situation.

The differential counterpart to the integral momentum equation (3.7) is
\[ \frac{1}{V(t)} \frac{\partial [V(t) \mathbf{u}]}{\partial t} + \nabla \cdot [\mathbf{u} (\mathbf{u} - \mathbf{b})] = -\nabla (p \mathbf{n}) + \nabla \cdot \mathbf{T} \quad (5.5) \]
\[ \frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \mathbf{u}) = -\nabla (p \mathbf{n}) + \nabla \cdot \mathbf{T} + \nabla \cdot (\mathbf{u} \mathbf{b}). \quad (5.6) \]

The momentum equation may be solved using the fractional step method outlined in Mahesh et al. (2004). Let the cell-centered fluid Cartesian velocities be \( u_i \) and the face-normal face center fluid velocity be \( v_n \). In addition for incompressible flow, we always have
\[ \nabla \cdot \mathbf{u} = 0. \quad (5.7) \]
Clearly, (5.6) and (5.7) are essentially the same as the incompressible momentum and continuity equations in Mahesh et al. (2004). The original NL term now becomes two terms:
\[ \nabla \cdot (\mathbf{u} \mathbf{b}) \to \text{CONV(LI)}, \quad \nabla \cdot (\mathbf{u} \mathbf{u}) \to \text{CONV(NL)} \quad (5.8) \]
in (5.6). In addition, evaluation of geometric quantities for the spatial differential operators now changes with time due to grid motion. Thus, the original fractional step method of Mahesh et al. should also be applicable. We may repeat their procedures as follows:
\[ \frac{\hat{u}_i - u^n_i}{\Delta t} = \text{CONV(NL)} + \text{VISC} + \text{CONV(LI)}. \quad (5.9) \]
The computed values of \( \hat{u}_i \) are used to obtain the face-normal velocities
\[ \hat{v} = \left( \frac{\hat{u}_i^{icv1} + \hat{u}_i^{icv2}}{2} \right) n_i, \quad (5.10) \]
where the face-normal, \( \mathbf{n} \) and \( \hat{v} \) point from the volume \( icv1 \) to \( icv2 \). The predicted face-normal velocities are projected using
\[ \frac{v_n - \hat{v}}{\Delta t} = -\frac{\partial p}{\partial n}. \quad (5.11) \]
Regardless of the motion of the grid, the divergence free constraint requires that
\[ \sum_{\text{faces of } cv} v_n A_f = 0, \quad (5.12) \]
\[ \Delta t \sum_{\text{faces of } cv} \frac{\partial p}{\partial n} A_f = \sum_{\text{faces of } cv} \hat{v} A_f. \quad (5.13) \]
Once the above Poisson equation (5.13) is solved, the Cartesian velocities are updated as
\[ \frac{u_i^{n+1} - \hat{u}_i}{\Delta t} = -\frac{\partial p}{\partial x_i}. \quad (5.14) \]
Hybrid simulation of helicopter flow

In a non-staggered formulation, pressure gradient at cell center may be evaluated using divergence theorem:

\[
\frac{\partial p}{\partial x_i} = \frac{1}{V} \sum_{\text{faces of cv}} p_{\text{face}} A_f n_i. \tag{5.15}
\]

But this leads non-conservative contribution to the discrete kinetic energy. If

\[
p_{\text{face}} = \frac{p_{\text{icv}} + p_{\text{nbr}}}{2},
\]

then we have

\[
\frac{u_i^{n+1} - \dot{u}_i}{\Delta t} = \frac{1}{V} \sum_{\text{faces of cv}} \left( \frac{p_{\text{icv}} + p_{\text{nbr}}}{2} \right) A_f n_i. \tag{5.17}
\]

From vector calculus contribution of pressure gradient term to kinetic energy is

\[
(u \cdot \nabla)p = \nabla \cdot (pu) - p(\nabla \cdot u). \tag{5.18}
\]

Summation over the volumes gives

\[
\sum_{\text{volumes}} u_i \frac{\partial p}{\partial x_i} V_{CV} = \sum_{\text{volumes}} \sum_{\text{faces of cv}} \frac{p_{\text{icv}} + p_{\text{nbr}}}{2} v_n A_f + \sum_{\text{volumes}} \sum_{\text{faces of cv}} p_{\text{icv}} v_n A_f. \tag{5.19}
\]

Because the second term on the right-hand-side of (5.19) is zero, we have

\[
\sum_{\text{volumes}} u_i \frac{\partial p}{\partial x_i} V_{CV} = \sum_{\text{volumes}} \frac{p_{\text{icv}}}{2} \sum_{\text{faces of cv}} v_n A_f + \sum_{\text{volumes}} \sum_{\text{faces of cv}} \frac{p_{\text{nbr}}}{2} v_n A_f. \tag{5.20}
\]

The first term on the right-hand-side of equation (5.20) is zero but not the second term. Thus, the pressure-gradient term is not conservative in its contribution to discrete kinetic energy in an ordinary non-staggered formulation with (5.15).

This problem can be corrected as follows. The contribution of pressure gradient to discrete kinetic energy is conserved if

\[
\frac{\partial p}{\partial n_{\text{face}}} = \frac{1}{2} \left( \left( \frac{\partial p}{\partial x_i} \right)_{\text{icv}} + \left( \frac{\partial p}{\partial x_i} \right)_{\text{nbr}} \right) n_i. \tag{5.21}
\]

This target requirement can be modified in a least-square sense by minimizing

\[
\sum_{\text{faces of cv}} \left\{ \left( \frac{\partial p}{\partial x_i} \right)_{\text{icv}} - \left( \frac{\partial p}{\partial n} \right)_{\text{face}} \right\}. \tag{5.22}
\]
In this instance,
\[
\sum_{\text{volumes}} u_i \frac{\partial p}{\partial x_i} V_{cv} = \sum_{\text{volumes}} \sum_{\text{faces of } cv} \frac{\partial p}{\partial n_{\text{face}}} v_n A_f
\]
\[
= \sum_{\text{volumes}} \sum_{\text{faces of } cv} \frac{1}{2} \left\{ \left( \frac{\partial p}{\partial x_i} \right)_{icv} + \left( \frac{\partial p}{\partial x_i} \right)_{nbr} \right\} n_i v_n A_f
\]
\[
= \sum_{\text{volumes}} \frac{1}{2} \left( \frac{\partial p}{\partial x_i} \right)_{icv} \sum_{\text{faces of } cv} n_i v_n A_f + \sum_{\text{volumes}} \frac{1}{2} \left( \frac{\partial p}{\partial x_i} \right)_{nbr} \sum_{\text{faces of } cv} n_i v_n A_f
\]
\[
= 0. \quad (5.23)
\]

The discrete kinetic energy conservation property in the unstructured fractional step method for incompressible flows developed by Mahesh et al. has been shown to be preserved when the mesh is undergoing solid-body rotation and translation with respect to an inertial reference frame. The convective contribution to discrete kinetic energy is conservative principally due to the properties of incompressibility and vanishing divergence of the grid velocity. The pressure gradient contribution term differs little to the situation considered by Mahesh et al. (2004).

6. Validations with diffuser and Couette flows

The unstructured LES solver CDP was converted into an unsteady RANS solver through incorporation of the V2F turbulence model of Durbin (1995). The implementation was validated using flow through the Obi planar diffuser (Obi et al. 1993). The Obi diffuser has an asymmetric planar configuration with a total expansion ratio of 4.7 and a single sided deflection wall of 10°. The inlet is a fully developed turbulent channel flow with a Reynolds number 500 based on friction velocity and channel half height. CDP RANS computation used a mesh of 360(x) × 80(y), and the results are compared with the experimental data of Buice & Eaton (1997) in Fig. 1. Also shown in the figure are CDP LES results of Wu et al. (2006) using a mesh of 590(x) × 100(y) × 110(z).

Moving mesh capability was added to CDP. Validation of the implementation was made using the laminar Couette flow. Two tests were performed in which the mesh rotates at the inner cylinder wall speed, and flow velocities in the inertial coordinate system are solved for. The radius of the inner and outer cylinders is 0.1 and 1.0, respectively. In the first case, the inner cylinder rotational speed is 10, and the outer cylinder remains stationary. In the second case, the outer cylinder rotates at one half the speed of the inner cylinder. As indicated by the results shown in Fig. 2, good agreement with theoretical results has been obtained.

7. Investigation on the interface conditions with SUmb–SUmb coupling

Coupling two different solvers requires a mutual exchange of information at interfaces. Since we consider the coupling of fully compressible and incompressible solvers, in which the incompressible code cannot provide all the information required for the fully compressible one, the best choice for the exchanged variables should be determined first. For
Figure 1. $\langle \overline{u} \rangle / u_b$ versus $y$ in the Obi diffuser. Upper: CDP RANS with V2F model; lower: CDP LES of Wu et al. (2006); symbols are the experimental data of Buice & Eaton (1997). ○ $x = 5.18$, • $x = 11.96$, ○ $x = 27.1$, + $x = 33.86$.

this purpose, SUnb–SUnb coupling was conducted for a laminar Taylor vortex convecting in a uniform flow (Iourokina & Lele 2005). This is an extension of the study by Kim et al. (2004) with a steady turbulent pipe flow.

In this study, two instances of SUnb take the upstream ($-9 < x < 7.7$) and downstream ($0 < x < 16.7$) parts of the entire computational domain, respectively. There is an overlap region of $0 < x < 7.7$ between the two domains, where the two solutions are expected to be almost identical without any special treatment. We considered the following three different interface conditions:

(a) Case 1: All five variables ($\rho, u, v, w, p$) are exchanged and imposed at both the exit and inlet of the upstream and downstream domains, respectively.

(b) Case 2: At the inlet of the downstream domain, $\rho, u, v$ and $w$ from the upstream domain are imposed, but $p$ is extrapolated using the Riemann invariant. At the exit of the upstream domain, only $p$ from the downstream domain is imposed, and all the other variables are extrapolated.

(c) Case 3: At the inlet of the downstream domain, all five variables are imposed. At the exit of the upstream domain, $\rho, u, v$ and $w$ from the upstream domain are imposed, but $p$ is extrapolated using the Riemann invariant.

Note that only Case 2 satisfies the characteristic theory. Figure 3 shows contours of the vertical velocity at the instant when the vortex enters the downstream domain. For
Figure 2. Azimuthal velocity profiles for laminar cylindrical Couette flows using moving-mesh capability of CDP: — CDP; ○, analytic solution for \((\Omega_i, \Omega_o) = (0, 10)\); ●, analytic solution for \((\Omega_i, \Omega_o) = (5, 10)\).

Figure 3. Contours of the vertical velocity at the instant when the vortex enters the downstream domain: Case 1 (left); Case 2 (middle); Case 3 (right). Shown are the results for the single SUmb (top), upstream (middle) and downstream (bottom) SUmb simulations.

comparison, the result of a single SUmb simulation over the entire domain was shown in this figure.

For all three cases, the vortex smoothly passes through the interface, and no evident numerical instability is found at the interface. Similar behaviors were also found at the instant when the vortex leaves the upstream domain. This indicates that coupled simulations are similar to the classical domain decomposition, and the characteristic theory
Figure 4. Convergence histories: , single S Umb simulation; and ○, Case 1; and ○, Case 2; and △, Case 3. Lines and symbols denote the upstream and downstream S Umb’s, respectively.

Figure 5. Velocity profile for $\epsilon = 0.1$: , smoothed profile; , original profile (Hardin’s solution).

does not necessarily have to be obeyed for the interface treatment. Figure 4 shows the convergence history for the three cases. An interface treatment obeying the characteristic theory (Case 2) shows a slightly better convergence rate, even though the difference is not so significant. Based on this result, we used an interface treatment in all the test cases described below such that all the velocity components and turbulence variables ($k$, $\epsilon$, $v^2$ and $f$) are exchanged and imposed at both S Umb and CDP interfaces, whereas the density is further assumed to be constant and the pressure is extrapolated using the Riemann invariant at the S Umb interface.

8. Validation with helical vortex

An accurate resolution of the tip vortex structure and the capturing vortex over long distances is important for modeling the aerodynamic performance of rotors in hover and
Figure 6. Iso-surface of the initial vorticity magnitude, $|\vec{\omega}| = 5$

Figure 7. Contours of vorticity magnitude at 4 instants. Upper left: $t = 0$; upper right: $t = 1.75$ (1 revolution); lower left: $t = 3.5$ (2 revolutions); lower right: $t = 5.25$ (three revolutions).

Figure 8. Time history of $\int \vec{\omega} dV$: ---, $z$ component; --., $y$ component; ---, $x$ component.
forward flight. Helical vortices model the tip vortex behind wind turbines, propellers and rotors in hover or in vertical flight. When the wake is fully developed sufficiently far from the blades, the tip vortex can be analyzed as an infinite helical vortex with constant radius and pitch.

Helical vortex filament is one of the small number of vortex geometries that can translate without deformation. Hardin (1982) derived a closed form solution for the velocity field induced by a helical vortex filament. To be able to perform numerical simulation, we need to study a helical vortex tube with small cross section. This is done by introducing a smooth cut-off function $\zeta_\epsilon$, see Cottet & Koumoutsakos (2000), that satisfies $\int_{-\infty}^{\infty} \zeta_\epsilon(\vec{x})d\vec{x} = 1$, and by taking the modified vorticity as the convolution of the delta vorticity distribution of the helical filament and the smooth cut-off function $\zeta_\epsilon$. The velocity field induced by the modified vorticity field can be calculated using Biot-Savart law. This divergence-free velocity field is used as the initial condition for the numerical simulation. Note that the singularity of the Biot-Savart integral has been removed by the smooth cut-off function and that the velocity field can be viewed as the convolution of the original vorticity field corresponding to the helical vortex filament and the smoothed kernel. On the other hand,

$$\vec{u}_\epsilon = \vec{K}_\epsilon \star \vec{\omega} = \vec{K} \star \zeta_\epsilon \star \vec{\omega} = \vec{K} \star \vec{\omega}_\epsilon,$$

(8.1)

where $\epsilon$ represents the core size, $\vec{K}$ is the kernel of Biot-Savart integral, $\vec{\omega}$ is the vorticity distribution of the helical vortex filament, and $\vec{K}_\epsilon$, $\vec{u}_\epsilon$, $\vec{\omega}_\epsilon$, represent the modified kernel, velocity and vorticity fields, respectively.

Using the second-order exponential function $\zeta_\epsilon(\vec{r}) = \frac{1}{\epsilon^3} \frac{3}{4\pi} exp(-\frac{r^2}{\epsilon^2})$, see Beale & Majda (1985), the initial velocity and vorticity fields of a helical vortex with constant radius $r_0$ and pitch $p$ are given by

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure9.png}
\caption{Tip vortex computed from CDP RANS in the Bradshaw wing flow.}
\end{figure}
Figure 10. Contours of velocity component $w$ in the tip vortex region of Bradshaw wing at $x = 1.5$. Upper: experiment; lower: CDP from coupled simulation.

\[
\vec{u}_e = -\frac{r_0}{2} \int_{-\infty}^{\infty} f(|\vec{r}|, \epsilon) \vec{r} \times \vec{t} \, ds
\]

(8.2)

and

\[
\vec{\omega}_e = \frac{3r_0}{2\epsilon^3} \int_{-\infty}^{\infty} exp\left(-\frac{|\vec{r}|^3}{\epsilon}\right) \vec{r} \, ds,
\]

(8.3)
Figure 11. Schematic of Leishman hovering rotor test, from Martin et al. (2001).

Figure 12. SUmb and CDP computation domains used in the integrated simulation of the Leishman rotor flow. The inner most dark strip is the rotor blade.

where

\[ f(\sqrt{r^2 + \epsilon^2}) = \frac{1 - e^{-\left(\frac{\sqrt{r^2 + \epsilon^2}}{\epsilon}\right)^3}}{\left(\frac{\sqrt{r^2 + \epsilon^2}}{\epsilon}\right)^3}, \]

\[ \vec{r} = \vec{x} - \vec{x}_H, \]

\[ \vec{x}_H = [r_0 \cos(s), r_0 \sin(s), s], \]

\[ \vec{t} = [-\sin(s), \cos(s), \frac{1}{r_0}]. \]
Figure 13. Meshes used for SUnb (left) and CDP (right) in integrated simulation of the Leishman rotor flow.

Figure 14. CDP mesh overlaid on the computed Leishman tip vortex.

Note that all the equations are non-dimensionalized with the length scale $p_r = \frac{p}{2\pi}$ and velocity scale $\frac{r}{2\pi p_r}$.

Figure 5 shows velocity profile for both helical vortex filament (Hardin’s solution) and the smoothed field with $\epsilon = 0.1$. At distances at least one core radius away from vortex, the two solutions match each other. At $r = 0$, Hardin’s solution is singular, while the smoothed velocity profile goes to the translational velocity of the smoothed helical vortex.

To perform numerical simulation of a helical vortex tube with unity radius and pitch, a cylindrical domain of unity height and radius 4 is discretized with 10 grid points across the vortex core (based on radius). The unstructured flow solver CDP is initialized with the initial velocity field in (8.2) with $\epsilon = 0.1$. Figure 6 shows one period of the iso-surface of the initial vorticity magnitude. By using the low-dissipative algorithm in CDP, we were able to preserve the vortex for more than three revolutions. Figure 7 shows the
time history of vorticity magnitude contours along the plane $x = 0$ for the first three revolutions.

By integrating the vorticity equation, one can obtain that for helical vortex, $\int \omega dV$ is constant. Figure 8 shows the time history of the volume integral of vorticity. It is clear that CDP preserves this quantity fairly well.

9. Validation on flow past Bradshaw wing

SUmb and CDP were applied separately as well as jointly in predicting the near-field behavior of a wingtip vortex flow, the Bradshaw wing. The computational domain included a half-wing with a NACA 0012 airfoil section, which also has a rounded wing
tip (Fig. 9). The wing has an aspect ratio of 0.75 and is mounded inside a wind tunnel at 10 degree angle of attack. The flow is treated as turbulent throughout the computational domain with a Reynolds number of $4.6 \times 10^6$ based on chord length and upstream velocity. The standard V2F turbulence model was used in the simulations. The origin of the coordinate system is located at 0.25 chord downstream of the leading edge. Experimental data on this flow are documented in Chow et al. (1997). In the SUmb and CDP coupled simulation, the flow upstream of $x = 1.25c$ was computed using SUmb and the flow downstream of $x = 1.0c$ computed using CDP. The exit is located at $x = 2.7c$. Figures 9 and 10 show that the wingtip vortex has been captured in the computation, and there is good agreement between the calculations and the experiment.

10. Validation on Leishman hovering rotor flow

Coupled SUmb and CDP simulations of the Leishman hovering rotor flow (Martin et al. 2001) are currently being performed. In the experiment, the rotor was tested at the hovering state in a specially designed flow conditioned cell. Corrections accounting for aperiodicity levels in the rotor wake (wandering) were made by Martin et al. (2001) on their measured experimental data using the procedure of Devenport et al. (1996). The tested rotor is one-bladed with a rectangular tip shape of a NACA 2415 airfoil section. The blade is 9.124 chord ($c$) in radius (Fig. 11). The tip Mach number and chord Reynolds number is 0.26 and 272,000, respectively. One of the primary challenges in computing the rotor flow is the substantial disparity in length scales relevant to the problem. Experiments have shown the diameter of a rotor vortex is approximately 0.05 to 0.1$c$. Various numerical tests also indicate that at least 10 mesh points across the vortex diameter are required in order to reasonably resolve such a tip vortex. Thus, there is a ratio of more than 1000 : 1 between the blade radius and the smallest grid spacing. In a typical LES channel flow calculation, the ratio between channel half height and one wall unit is the same as the frictional velocity Reynolds number.

Figure 12 shows the computational domains for SUmb and CDP in the coupled simulation with the adoption of an overlapping region. SUmb's inner boundary is the blade.

![Figure 17. Swirl velocity of tip vortex as a function of vortex radial coordinate at wake age 25°. Solid line: CDP solution from the integrated simulation; o measured data of Martin et al. (2001); • corrected data of Martin et al. (2001).](image-url)
surface, and its outer boundary is the irregular surface shown in the figure. CDP’s outer boundary is the cylindrical tank and its inner boundary is the rectangular brick in Fig. 12. Flow conditions at the interfaces are exchanged between the two solvers in a time accurate fashion. Far field boundary condition used in CDP is from Conlisk (1997), derived using one-dimensional momentum theory and mass conservation. Substantial mesh concentration is applied in the tip vortex region (Fig. 13). For example, there are 250 points distributed radially in CDP from $7.5c$ to $10c$. Figure 14 indicates that there are approximately 10 points across the core of the tip vortex, and the mesh is nearly uniform in the vortex cross-sectional plane. The total number of control volumes in SUmb is 3.2 million and 8 million in CDP. At 3 degrees of wake age, there is very good agreement between SUmb calculation and the experiment, see Fig. 15. Circumferentially, the CDP interface is located at approximately 6 degrees downstream of the blade trailing edge. Results from both CDP are SUmb at this station are shown in Fig. 16. Since there are no experimental data available at this station, the data at 3 degrees wake age were plotted on the figure. This is useful, because comparison of experimental data at 3 degree and 25 degree (Fig. 17) suggests that there must be small changes between data at 3 and 6 degrees. Figure 16 indicates that there is already a decay at this location in the SUmb solution. Based on these results, an improved CDP mesh has been generated. In the refined CDP mesh, there are 400 points distributed radially in CDP from $7.5c$ to $10c$, and the CDP interface is moved closer to the trailing edge at about 3 degrees downstream of the blade trailing edge. Calculations using the improved mesh are currently being performed.

Acknowledgment

This work is supported by Helicopter Quieting Program from the Defense Advanced Research Projects Agency (DARPA) of the United States Department of Defense.

REFERENCES


HAM, F. & IACCARINO, G. 2004 Energy conservation in collocated discretization


