Simulation of Unsteady Rotor-Fuselage Interactions Using Unstructured Adaptive Meshes

Young Min Park  Hwa Jin Nam  Oh Joon Kwon
Graduate Research Assistant  Associate Professor

Department of Aerospace Engineering
Korea Advanced Institute of Science and Technology
373-1 Guseong-dong, Yuseong-gu, Daejeon 305-701, KOREA
ojkwon@mail.kaist.ac.kr

A three-dimensional parallel Euler flow solver has been developed for the simulation of unsteady rotor-fuselage interaction aerodynamics on unstructured meshes. To simulate the unsteady rotor wake effectively, the flow field is divided into a moving zone rotating with the blades and a stationary zone containing the fuselage. A sliding mesh algorithm is applied for the convection of flow variables across the cutting boundary between the two zones. Quasi-unsteady mesh adaptation is adopted to enhance the spatial accuracy of the solution. Mesh deformation due to blade motion in forward flight is handled by using the spring analogy and cell edge-collapsing. Validation of the rotor-alone configuration was made for the AH-1G rotor in forward flight. The rotor-fuselage interaction study was made for flow around the Georgia Tech configuration and the ROBIN fuselage. It is shown that the present method is efficient and robust for the prediction of complicated unsteady rotor-fuselage interaction phenomena.

Introduction

Prediction of the flow field of helicopter rotors in forward flight is regarded as one of the most challenging problems in current computational fluid dynamics. This is mainly due to the unsteadiness of complex rotor wake and the asymmetric nature of rotor disk loading. Addition of fuselage underneath the rotor further amplifies this complexity, requiring to solve unsteady three-dimensional multiple bodies in relative motion. The mutual aerodynamic interference and the aerodynamic inharmoniousness between the rotor and the fuselage can bring about the undesirable noise, air loads and vibration, which can incur serious damage on the helicopter. Therefore, for the stable operation of helicopter, elaborate aerodynamic analysis of the rotor-fuselage interaction must be preceded.

Traditionally, singularity methods such as lifting line/free-wake model have been used to model the rotor blades and fuselage. Owing to the development of high performance computers, analysis of rotor-fuselage interaction became possible using Euler and Navier-Stokes methods. However, these methods were based on the momentum source for modelling the rotor and are good only for solving time-averaged flow. An alternative way of predicting the time-averaged rotor-fuselage interaction was also proposed by solving the Navier-Stokes equations on overset grid and modelling the rotor as an actuator disk. This actuator disk model has also been adopted on unstructured meshes. Extension of the actuator disk model was made to simulate unsteady inflow and fuselage surface pressure by combining the Generalized Dynamic Wake Theory (GDWT) with an overset grid Navier-Stokes flow solver. An unsteady source distributed disk model coupled to a parallel Navier-Stokes solver was also used for the simulation of unsteady rotor-fuselage interactional aerodynamics.

Even though the above-mentioned methods based on momentum source or actuator disk model turn out to provide reasonable time-averaged/time-accurate solutions for rotor-fuselage interaction problem, limitations still exist for capturing realistic unsteady flow features such as tip-vortex generation, blade-vortex interaction, or tip-vortex impingement on fuselage. The realistic flow modelling has been made using overset structured grid methods, which easily describe the relative motion between rotor blades and fuselage. The overset grid method was also applied to solve rotor-fuselage interaction aerodynamics of the half and full V-22 Osprey configuration.

CFD method based on unstructured meshes has an advantage of handling complex geometry and can easily improve the solution accuracy by refining cells locally as required. Application of the unstructured
adaptive mesh technique for solving Euler/Navier-Stokes flows around rotor-alone configuration in hover has been successfully accomplished by several researchers.\textsuperscript{16–20} The unstructured mesh method was also applied for solving the propeller-wing-body configuration by adopting a rotating mesh for the propeller.\textsuperscript{21} Recently, an unstructured mesh method was developed by the present authors for the efficient simulation of unsteady flow fields around the helicopter rotors in hover and in forward flight using a quasi-unsteady adaptive sliding mesh technique.\textsuperscript{22} However, the unstructured mesh method has not been applied to unsteady helicopter rotor-fuselage interaction aerodynamics.

In the present study, the previous unstructured adaptive mesh method\textsuperscript{22} has been extended for the simulation of unsteady rotor-fuselage interaction. For this purpose, a parallel unsteady Euler flow solver has been developed. The relative motion between the rotor blades and the fuselage is enabled by dividing the flow domain into two zones. The upper rotating zone includes the rotor blades and the near wake. The lower stationary zone contains the fuselage and the far wake. The accuracy of solution and rotor wake capturing is enhanced by adopting a quasi-unsteady solution-adaptive mesh refinement technique. Validation of the rotor-alone configuration was made for the AH-1G rotor in forward flight.\textsuperscript{23, 24} The rotor-fuselage interaction study was made for flow around the Georgia Tech configuration\textsuperscript{25, 26} and the ROBIN fuselage.\textsuperscript{27, 28} Time-averaged and unsteady flow field and fuselage surface pressure are compared with experiment for validation. Calculations were made on a Linux-based PC cluster having 2.4Ghz Pentium IV CPUs.

**Numerical Method**

**Spatial Discretization and Time integration**

The governing unsteady Euler equations can be written in finite-volume form on inertial coordinate system. The inviscid flux across each cell face is computed by using the Roe’s flux-difference splitting formula.\textsuperscript{29} To obtain second-order spatial accuracy, estimation of the state variables at each cell face is achieved by interpolating the solution using the Taylor series expansion in the neighborhood of each cell center. The cell-averaged solution gradient required at the cell center for the above expansion is computed from the Gauss’ theorem by evaluating the surface integral for the closed surface of the tetrahedra. This process can be simplified by using some geometrical invariant features of the tetrahedra.\textsuperscript{30} The expansion also requires the nodal value of the solution, which can be computed from the surrounding cell center data using a second-order accurate pseudo-Laplacian averaging procedure.\textsuperscript{31}

An implicit time integration algorithm based on the linearized second-order Euler backward differencing is used to advance the solution in time. The linear system of equations is solved at each time step by using a point Gauss-Seidel method.

**Boundary Condition**

At the far-field boundary, the pressure is fixed to the freestream value, and other flow variables are extrapolated from the interior. The far-field boundary is located at five radii away from the rotor center-of-rotation in all directions.

On the solid surface of rotor blades and fuselage, the flow tangency condition is applied.

**Parallel Implementation of Flow Solver**

The flow solver is parallelized by partitioning the global computational domain into local subdomains by using the MeTIS library.\textsuperscript{32} The local subdomain mesh data is allocated in each processor, and the calculation is performed for each subdomain by exchanging the updated solution across the subdomain boundary.

Initially, face values of the flow variables are exchanged across the boundary to calculate flux Jacobian at the subdomain boundary. Next, cell-center values are exchanged for Gauss-Seidel iteration. Boundary node values and weighting factors for Laplacian averaging are also communicated for high-order reconstruction. In the present study, cell data is exchanged at every sub-iteration, and face and node data is transferred for every outer iteration.

**Sliding Boundary Treatment**

In order to handle the complicated flow field effectively, the computational domain is decomposed into two zones. The upper zone contains the rotor blades and rotates with them. The stationary lower zone covers the rest of the flow field including fuselage and far wake of the rotor. The computational mesh is generated separately for each zone, and a sliding mesh algorithm is developed for the convection of flow variables across the boundary.\textsuperscript{22} Information about the overlapping area between upper and lower sliding boundary faces is used to calculate the flux at each boundary face such that geometric conservation is satisfied across the sliding boundary. Second-order spatial accuracy of the flux discretization is maintained across the sliding boundary by introducing ghost cells attached to each of the sliding boundary faces inside the opposite flow zone.

In order to avoid the complexity of data management across the sliding boundary, parallel domain partitioning is made for each zone separately. Time-varying information for data communication is retained in all processors for the complete sliding boundary. After each time step advancement, local boundary data updated for all processors is collected and redistributed to each processor by using an MPI subroutine, MPI\_ALL\_GATHER.
Mesh Adaptation

A mesh adaptation algorithm is adopted to reduce the numerical dissipation and to enhance the spatial accuracy of the solution. At every adaptation level, targeted cells are marked by testing the vorticity magnitude obtained from the current solution. The tagged cells are divided by adding new node points in the middle of six edges of each tetrahedral cell. Buffer cells are also used to preserve the connectivity between the divided cells and the surrounding cells.

In forward flight, position of the tip vortex trajectory changes continuously. Even though capturing the moving tip vortices in a time-accurate manner is possible by dynamically applying the adaptive mesh technique, the computational overhead required for frequent mesh refinement and coarsening becomes unreasonably large. In the present study, a ‘quasi-unsteady’ dynamic mesh adaptation is adopted to maintain proper mesh resolution, while avoiding excessive computational time required for the dynamic mesh adaptation in a time-accurate manner.

As the blades rotate, cells having high vorticity are marked at every time step. Once one period of rotor revolution is completed, calculation is paused and the mesh adaptation is applied for cells tagged during that period. Then, new domain decomposition is applied to the refined mesh for load balancing, and the calculation resumes. This procedure is repeated until a satisfactory solution is obtained. Using this adaptation procedure, computational time required for dynamic mesh refinement and coarsening can be significantly reduced.

Blade Motion and Mesh Deformation

Helicopter rotors in forward flight accompany periodic motion of the blades, which can be described by using the Fourier series as a function of the azimuth angle, $\psi$:

$$\beta(\psi) = \beta_0 + \beta_1 \cos(\psi) + \beta_2 \sin(\psi) + \ldots$$  
$$\theta(\psi) = \theta_0 + \theta_1 \cos(\psi) + \theta_2 \sin(\psi) + \ldots$$

where $\beta$ and $\theta$ represent the blade flapping and pitch angles, respectively.

This blade motion creates additional complexity in handling the mesh so that grid points surrounding the blade surface need to cope with the motion. In order to eliminate the difficulty associated with large tip flap deflection, the sliding boundary plane is set parallel to the tip path plane underneath the rotor. The mesh deformation due to blade pitching motion is treated by using the spring analogy with wall distance correction. An additional treatment is also made to get rid of highly skewed cells generated during mesh deformation by using cell edge-collapsing.

Rotor Trim

In order to match the calculated thrust to the desired level and to eliminate the rotor aerodynamic moments, a rotor trimming procedure is enforced in forward flight. The thrust and moment coefficients can be expressed as a function of collective and cyclic pitch angles.

$$C_T = C_T(\theta_0, \theta_{1c}, \theta_{1s})$$
$$C_{Mx} = C_{Mx}(\theta_0, \theta_{1c}, \theta_{1s})$$
$$C_{My} = C_{My}(\theta_0, \theta_{1c}, \theta_{1s})$$

(3)

The estimated correction angles of the control settings, $\Delta \theta_0$, $\Delta \theta_{1c}$ and $\Delta \theta_{1s}$, can be obtained by using the Newton-Rhapson iterative method.\(^{33}\)

$$\begin{pmatrix} \Delta \theta_0 \\ \Delta \theta_{1c} \\ \Delta \theta_{1s} \end{pmatrix} = \left( \begin{pmatrix} \frac{\partial C_T}{\partial \theta_0} & \frac{\partial C_T}{\partial \theta_{1c}} & \frac{\partial C_T}{\partial \theta_{1s}} \\ \frac{\partial C_{Mx}}{\partial \theta_0} & \frac{\partial C_{Mx}}{\partial \theta_{1c}} & \frac{\partial C_{Mx}}{\partial \theta_{1s}} \\ \frac{\partial C_{My}}{\partial \theta_0} & \frac{\partial C_{My}}{\partial \theta_{1c}} & \frac{\partial C_{My}}{\partial \theta_{1s}} \end{pmatrix} \right)^{-1} \begin{pmatrix} C_T^{desired} - C_T \\ -C_{Mx} \\ -C_{My} \end{pmatrix}$$

(4)

The trim solution requires many cycles before convergence, which is computationally very expensive. To reduce the computational time, this rotor trim procedure is performed using computational mesh without adaptation. Each trim cycle consists of seven rotations of the rotor, three for calculating the sensitivities and the rest for solution iteration.

Results and Discussion

Lifting Rotor in Forward Flight

The initial calculation was made for a lifting AH-1G rotor-alone configuration in forward flight to validate the present unsteady flow solver. The rotor has been tested and well documented data is available for comparison.\(^{23,24}\) The two-bladed teetering rotor has a rectangular planform shape and an aspect ratio of 9.8. The blade is linearly twisted by -10 degrees from root to tip. The flight measurement compared with the present calculation was made at a tip Mach number of 0.65, an advancing ratio of 0.19, and a time-averaged total thrust coefficient of 0.00464. The measured blade first harmonics are presented in table 1, which were used as an initial estimation for rotor trim.

Table 1 Blade harmonics for the AH-1G rotor (angles in degree).

<table>
<thead>
<tr>
<th></th>
<th>$\theta_0$</th>
<th>$\theta_{1s}$</th>
<th>$\theta_{1c}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flight Test [23,24]</td>
<td>6.0</td>
<td>-5.5</td>
<td>1.7</td>
</tr>
<tr>
<td>Yang et al. [33]</td>
<td>8.0</td>
<td>-6.5</td>
<td>2.5</td>
</tr>
<tr>
<td>Present Calculation</td>
<td>5.8</td>
<td>-4.9</td>
<td>1.25</td>
</tr>
</tbody>
</table>

Initial calculation was made on a coarse mesh containing 371,103 cells for the rotating upper zone and
112,574 cells for the lower stationary zone. After three levels of mesh adaptation, the number of cells increased to 1,473,019 and 1,575,007 for each zone, respectively. The number of surface triangles on the sliding boundary plane also increased from 8,118 to 43,524 for the upper zone and from 10,150 to 36,794 for the lower zone. The sliding boundary was set at one chord length underneath the rotor tip path plane.

The refined cells inside the flow domain after mesh adaptation are shown in Fig. 1 for both upper and lower zones. In the upper zone, cell refinement was made along the trace of the tip vortex trajectory, which resulted in a circular ring shape. Inside the lower stationary zone, refined cells are uniformly distributed below the rotor disk with a slight skewness in downward direction, which reflects the skewed wake pattern typical for lifting rotors in forward flight. This cell refinement pattern is also shown on the sliding boundary plane as small cells widely spread in the direction of freestream.

In Fig. 2, history of the thrust and moment coefficients are presented during rotor trim procedure. It is shown that the thrust approaches to the experimental value after three trim cycles (approximately 18 revolutions). However, the aerodynamic moments require approximately twice more rotor revolutions before convergence. The total time-averaged thrust coefficient after trim was 0.00462. The resultant harmonics are summarized in Table 1, and compared with other results. The predicted collective and cyclic pitch angles are slightly smaller than the experiment, which may be attributed to the inviscid flow nature of the present calculation. The inviscid torque remains almost constant throughout the trim procedure.

The predicted chordwise surface pressure distributions are compared with the flight test at 91% span in Fig. 3. The results of a hybrid method including aeroelastic blade effect are also compared in the figure. Fair comparison is made at all azimuthal positions of the blade. The predicted suction peak at the leading edge is consistently low on the advancing side, which is similar to the other calculation. The irregular behavior near the leading edge region observed by the other researcher is not shown on the present results. The predicted surface pressure on the retreating side compares well with the experiment.

The sectional thrust variations at several spanwise locations are compared with the flight test data and the results by other prediction in Fig. 4. It is shown that the present calculation compares well with the flight test for all spanwise locations. The blade-vortex interaction near 270 degree azimuth angle is well captured by using the present method. The weak blade-vortex interaction at 90 degree azimuth angle is also reasonably well captured compared to that of the other prediction.

**Georgia Tech Rotor-Fuselage Configuration**

The first validation of rotor-fuselage interaction was made for the configuration tested at Georgia Tech. This configuration has a two-bladed teetering rotor and a generic cylindrical fuselage. The blade has a rectangular planform shape with an aspect ratio of 5.3 and is made of an NACA0015 airfoil section. The rotor is operating at a blade tip Mach number of 0.295 and an advance ratio of 0.1. The blade collective pitch angle is set at 10 degrees, and the cyclic variation is not allowed. The rotor shaft has a front tilt of 6 degrees. The thrust coefficient from experiment is approximately 0.0091. The sliding boundary plane was located at 0.26 chord length below the rotor disk.

The initial coarse mesh contains 247,010 cells in the rotating zone and 230,400 cells in the stationary zone. After mesh adaptation, the number of cells increased to 1,782,134 and 1,623,794 for each zone. The number of sliding boundary faces also increased from 6,611 and 6,427 to 34,279 and 27,234. Figure 5 shows the surface mesh and the refined cells inside the flow domain for the Georgia Tech configuration after mesh adaptation. The rotor proceeds 0.2 degree for each time step, which requires approximately six hours to complete one rotor revolution for the refined mesh. Calculation was made using 15 and 13 processors for rotating and stationary zones, respectively.

Figure 6 shows the time-averaged downwash and streamwise velocity component distribution at 15% chord length above the fuselage. Good agreement was obtained between the present calculation and the experiment for both velocity components, except that the streamwise velocity is predicted higher near the leading-edge of the rotor disk. Two peaks at the front and at the rear part represent the presence of the tip vortex passage. The peak values are well predicted even though the location is slightly off from the experiment at the front. The results obtained after mesh adaptation provide slightly better resolution of the peaks, demonstrating the effect of mesh adaptation for capturing the wake.

In Fig. 7, time-averaged surface pressure distributions along the fuselage are presented at four circumferential locations. It is shown that good comparison is obtained between the present calculation and the experiment for all cases. Along the top of the fuselage, two distinct peaks are observed, which is due to the tip vortex impingement. The location and peak value are well predicted, even though the peak is slightly lower at the downstream. The consistently lower pressure at the advancing side is due to the higher velocity induced by the present solver. The bottom line of the fuselage is not significantly affected by the wake.

The time-averaged inflow distribution at four azimuthal positions of the rotor is shown in Fig. 8. The predicted results compare well with the experiment for all azimuth angles. The upwash at the front leading-
edge of the rotor disk is also well predicted. The effect of mesh adaptation becomes clear near the tip region of the rotor disk, showing rapid decrease in downwash induced by the strong tip vortex. The inboard regions are not affected as much.

Variation of the unsteady inflow observed at three fixed points located at 187.5° azimuth angle is presented in Fig. 9. The effect of blade bound vortex is clearly visible as large abrupt changes of inflow associated with the blade passage. The peak value of variation is larger at outboard due to higher loading. At the tip, the magnitude of peak becomes small due to the reduced loading. Additional secondary upwash is observed at this tip position, which is induced by the tip vortex from preceding blades located at the inboard of the tip under the blade. The overall agreement with experiment is good, even though the experimental data points are insufficient around the peak for comparison.

The instantaneous unsteady pressure distribution along the top of the fuselage is presented at four blade azimuth angles. In general, the predicted results compare well with the experiment for all fuselage locations. The effect of blade passage is well represented at the front region of the fuselage. However, the peak measured from the experiment at downstream associated with the tip vortex core impingement is not properly resolved in the present calculation, which suggests that further refinement of mesh is still desirable.

ROBIN Configuration

Second validation of the present method was made for ROBIN (ROtor Body INteraction) configuration tested at NASA Langley27–28. The rotor has four blades, which are made of an NACA0012 airfoil section and have a rectangular planform with an aspect ratio of 12.98. The blade is linearly twisted by -8 degrees from root to tip, and has a root cutout of 0.24R. The shaft is tilted forward by 3 degrees. The sliding boundary plane was set at 1.66 chord length underneath the rotor disk plane.

Among various measurement cases reported in the experiment, the one with a blade tip Mach number of 0.5 and an advance ratio of 0.151 is chosen for validation. The measured nominal collective pitch angle was 10.3 degrees at 75% span, giving the time-averaged thrust coefficient of 0.0063. The longitudinal and lateral cyclic pitch angles are -2.7 and 2.4 degrees, respectively. The converged solution for trim was obtained after 28 revolutions on the coarse mesh, which resulted in the collective and cyclic pitch angles of 6.5, -2.2, and 2.0 degrees, respectively. The calculated time-averaged thrust coefficient on the adapted mesh was 0.00627.

The coarse mesh used for initial calculation and rotor trim contains 617,278 and 547,372 cells for rotating and stationary zones, respectively. After two levels of mesh adaptation, the number of cells increased to 1,951,054 and 1,950,461. Calculation was made using 15 processors for each zone, which takes approximately 7 hours of elapsed CPU time for one rotor revolution on the refined mesh. At each time step, 0.2 degree of rotor rotation was made. Figure 11 shows the surface mesh distribution on the rotor blades and the fuselage, and the refined cells inside the flow domain near the symmetric cutting plane.

The time-averaged pressure distribution around the fuselage at selected streamwise locations is compared with experiment in Fig. 12. Good comparison is made at X/R=0.35 located ahead of the pylon. At two locations downstream of the pylon, the overall agreement is fair. There exists some deviation between the predicted pressure and the experiment, particularly at negative z-locations. These points are under the fuselage, and accurate prediction cannot be made using the present inviscid flow solver.

In Figs. 13 and 14, time-averaged downwash and streamwise velocity distribution at 1.15 chord length above the rotor disk are presented. Each velocity component is normalized by the tip Mach number. Good agreement in downwash velocity distribution is obtained between the present calculation and the experiment for both front-to-rear and left-to-right directions. The upwash at the leading-edge of the rotor disk is also predicted very well. Even though the predicted streamwise velocity is slightly higher than that of the experiment, the overall agreement is still good for both its magnitude and slope of variation in both directions.

In Fig. 15, predicted unsteady pressure variation is compared with experiment on four selected fuselage surface points. Along the topline of the fuselage, the peak-to-peak is slightly overpredicted. The predicted results also show approximately 30 degree phase shift from the experiment. At the advancing side of the fuselage, the magnitude of variation is significantly smaller than the experiment. A similar phase difference is also observed on the retreating side of the fuselage. Even though mesh adaptation was applied to enhance the spatial accuracy of the solution, these discrepancies in magnitude and phase may still be attributed to the insufficient mesh resolution for capturing the far wake of the rotor.

Figure 16 shows the predicted and measured unsteady induced inflow variations at a fixed point above the rotor. The experimental data was obtained for an isolated rotor used for ROBIN configuration. The results are also compared with those based on GDWT theory.9 The peak-to-peak of the out-of-plane inflow is slightly underpredicted. It is shown that the present result is almost identical to the prediction based on GDWT theory in both directions.

In Fig. 17, instantaneous vorticity contours on a vertical cutting plane at the rear edge of the rotor disk are presented. It is shown that there exist two distinct trailing vortex formations at both sides of the
rotor disk. Layers of skewed wake vortex, generated from preceding blades and convected downstream, are also shown underneath the rotor. Convection of flow across the sliding boundary is well confirmed in the figure, demonstrating the validity of the present sliding boundary algorithm.

Conclusions

A parallel inviscid flow solver has been developed for the calculation of unsteady rotor-fuselage interaction aerodynamics using unstructured meshes. For the efficient simulation of unsteady wake, flow domain is decomposed into two zones. The upper zone contains the rotor blades and rotates with them. The lower zone is stationary and covers the fuselage and rotor wake. A sliding boundary algorithm is applied for the proper convection of flow variables and to satisfy the conservation property across the boundary between the two zones. A 'quasi-unsteady' adaptive mesh technique is used to enhance the spatial accuracy of the solution. A deforming mesh algorithm based on the modified spring analogy is adopted to handle the blade motion and rotor trim. Calculation was made for a lifting rotor-alone configuration for validation. The method was applied to rotor-fuselage configurations tested at Georgia Tech and NASA Langley. It was demonstrated that the present method is efficient and robust for simulating unsteady rotor-fuselage interaction.

References


Fig. 1 Refined cells inside the flow domain for AH-1G rotor in forward flight.

Fig. 2 Trim history of thrust and moment coefficients for AH-1G rotor in forward flight.

Fig. 3 Chordwise surface pressure distributions of AH-1G rotor at 91% spanwise section.

Fig. 4 Sectional thrust variations of AH-1G rotor blade in forward flight.
Fig. 5 Computational mesh after adaptation for Georgia Tech configuration.

Fig. 6 Time-averaged velocity components at 0.15c above the fuselage for Georgia Tech configuration.

Fig. 7 Time-averaged pressure distribution on the fuselage for Georgia Tech configuration.
Fig. 8 Time-averaged inflow velocity distribution at 0.15c above the rotor disk for Georgia Tech configuration (positive downward).

Fig. 9 Unsteady inflow variation observed at fixed locations 0.15c above the rotor disk for Georgia Tech configuration (positive downward).

Fig. 10 Instantaneous surface pressure distribution along the top of the fuselage for Georgia Tech configuration.
Fig. 11 Surface triangulation and refined cells inside flow domain for ROBIN configuration.

Fig. 12 Time-averaged fuselage surface pressure distribution for ROBIN configuration.

Fig. 13 Time-averaged downwash distribution for ROBIN configuration at 1.15c above the rotor disk.

Fig. 14 Time-averaged streamwise velocity distribution for ROBIN configuration at 1.15c above the rotor disk.
Fig. 15  Unsteady pressure variation on selected fuselage surface points.

Fig. 16  Induced inflow in two directions at $r/R = 0.8$, $\psi = 84^\circ$, and 1.7c above the rotor disk.

Fig. 17  Vorticity contours on a cutting plane at $x/l = 1.37$. 