The effect of flapping frequency on the aerodynamics of NACA0012 wing

Marcel Ilie[1] and Geoffrey Sullivan[2]
Georgia Southern University, Statesboro, GA, 30458

Computational studies of transient three-dimensional flapping wing are carried out using the large eddy simulation (LES) approach. The studies concern the understanding of the effect of flapping motion on the aerodynamic performance and aeroacoustics noise of a finite NACA0012 wing. The flapping motion of the wing generates trailing-edge vortices whose magnitude and scale vary in time. The near-field flow is associated with periodic flow separation and reattachments, which cause a time-dependent aerodynamic coefficients.

I. Introduction

Despite the simple geometry, the flow past a wing exhibits complex flow phenomena that may involve transition to turbulence (inside the boundary layer), boundary layer separation, recirculation bubbles, and geometry-dependent vortex flows in the wake region, resulting in a multifaceted and challenging flow field. These flow phenomena are very common in many aerodynamic applications, such as an aircraft at high angle of attack, scramjet inlet flow, etc.
Flutter is a common phenomenon that the wing or tail of an aircraft may encounter in turbulent flows. Therefore, the flutter phenomenon may augment the flow separation, tip-vortex formation and wake.
The motivation for the present investigation is driven by the desire for a better understanding of the flow separation and dynamic stall associated with the aircraft wing undergoing the flutter phenomenon.
Recent studies concerning flow transition employed a turbulence/transition region model that uses a turbulence kinetic energy equation in conjunction with algebraic length scale relationships. The model can predict the location of transition as well as simulate the transition region. The model can also account for re-laminarization in strong favorable pressure gradients. Comparisons with many experiments are presented with good results.
The process of transition from laminar to turbulent flow is very complicated. It was observed that transition starts as turbulent spots which form in the laminar region and propagate downstream, eventually coalescing to form the turbulent region. Researchers developed formulas using this spot theory to compute the intermittency of the transition region, where the intermittency is defined to be the percentage of time that a flow is turbulent at a given location.

The use of RANS methods, significantly rely on turbulence models to capture all the relevant turbulence scales. RANS methods predict the noise using the mean flow properties. Due to the fact that the flow transition is a multi-scale problem, involving a wide range of length and time scales, the use of RANS-based prediction methods remains limited. Although RANS methods are useful for predicting the aerodynamic coefficients, holding accurately up to some extent, they are usually not suitable or reliable for an accurate prediction of flow separation.

The recent improvements in the processing speed of computers make the applicability of Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES) to turbulent flows more feasible. However, due to a wide range of length and time scales present in turbulent flows, the use of DNS is still limited to low-Reynolds-number flows and relatively simple geometries. It is known that the number of grid points required for DNS is proportional to $Re^{9/4}$. Direct Numerical Simulation of high-Reynolds number flows of practical interest would necessitate high resolution grid requirements that are far beyond the capability of the most powerful computers available now days. In order to overcome the grid requirements issues, turbulence has to be modelled in order to perform simulations for

---

[1] Assistant Professor, Department of Mechanical Engineering,
[2] Undergraduate Student, Department of Mechanical Engineering

problems of practical interest. Large Eddy Simulation, with a lower computational cost, is a promising alternative method to DNS, for simulations of high Reynolds-number flows. LES methods are capable of simulating flows at high Reynolds number, LES method being independent of Reynolds number. In Large Eddy Simulation, the large scales are directly solved, while the small scales are modelled. Since flow transition is an unsteady process, LES is probably the most affordable computational tool to be used, since it is the only way, other than DNS, to obtain a time-accurate unsteady solution.

The paper is organized in the following manner. Section 2 provides an overview of the governing equations and numerical methods used in solving the simulation. Subsequently, section 3 describes the computational model and mesh generation. The main section, section 4, presents the results and discusses the implication of the simulation analysis. Section 5 concludes the research findings and makes suggestions for future work.

II. Governing equations and numerical method

A. Large Eddy Simulation

Large Eddy Simulation is a result of space-filtering operation applied to the Navier – Stokes equations. The filtered equations are:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\overline{u}_i \overline{u}_j\right) = -\frac{\partial \overline{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} + \frac{1}{\mathrm{Re}}\frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} \tag{2}$$

where τ_{ij} is the subgrid scale stress (SGS) given by:

$$\tau_{ij} = \overline{u_i u_j} - \overline{u}_i \overline{u}_j \tag{3}$$

and is modeled, where $\overline{u_i}$ is the velocity component of the resolved scales, $\overline{p}$ is the corresponding pressure and Re is the Reynolds number. The filtering procedure provides the governing equations for the resolvable scales of the flow field. Although the continuity equation (1) of the resolved quantities is equal to the original unfiltered one, the filtered momentum equation (2) includes an additional term for the non-resolvable subgrid scale stresses τ_{ij}, which results from filtering the non-linear convective fluxes. The role of τ_{ij} is to describe the influence of the small–scale structures on the larger eddies.

The Large Eddy Simulation concept leads to a closure problem similar to the one obtained by the Reynolds-averaged approach. Hence it is possible to classify the turbulence models starting with zero-equation models and ending up with Reynolds stress models. The non-resolvable small-scale turbulence in LES is much less problem-dependent than the large–scale turbulence, such that the subgrid scale turbulence can be represented by relatively simple models such as zero-equation eddy-viscosity models. The Smagorinsky model is an eddy viscosity model based on the Boussinesq approach, assuming that in analogy to the viscous stresses in laminar flows, the turbulent stresses are proportional to the mean velocity gradients or in a general sense to the large scale strain rate tensor $\overline{S}_{ij}$:

$$\tau_{ij} - \frac{1}{3}\delta_{ij}\tau_{kk} = 2\nu_T \overline{S}_{ij} \tag{4}$$

In the present work the SGS proposed by Smagorinsky and Lilly is used. In this analysis the value of Smagorinsky constant was set to 0.1. The SGS stresses are related to the strain rate tensor by SGS viscosity, ν_T:

$$\tau_{ij} - \frac{1}{3}\delta_{ij}\tau_{kk} = 2\nu_T \overline{S}_{ij} \tag{5}$$

The SGS viscosity ν_T is given by:

$$\nu_T = \rho (C_s D_{wall} \Delta)^2 \left| \overline{S} \right| \tag{6}$$

where C_s is the Smagorinsky constant, D_{wall} represents the van Driest wall damping factor, Δ is the filter width and $\left| \overline{S} \right|$ represents the magnitude of the large-scale strain-rate tensor.

$$\overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_j} \right) \tag{7}$$

$$\left(\overline{S_{ij}}\,\overline{S_{ij}} \right)^{\frac{1}{2}} = \left| \overline{S} \right| \tag{8}$$

The subgrid length l is assumed to be proportional to the filter width $\overline{\Delta}$:

$$l = C_s \overline{\Delta} = C_s \left(\Delta_x \Delta_y \Delta_z \right)^{\frac{1}{3}} \tag{9}$$

The filter width $\overline{\Delta}$ is correlated with a typical grid spacing given by the cube root of the cell volume. Taking into account the reduction of the subgrid length l near solid walls, the length scale is usually multiplied by a Van driest damping function, as

$$l = C_s \overline{\Delta} [1 - \exp(-y^+ / 25)^3]^{0.5} \tag{10}$$

Although theoretical values of $C_s \approx 0.16$ for homogenous, isotropic turbulence can be found in literature, smaller values are usually applied in LES computations of non-homogenous and non-isotropic flows. All computations in the present work were carried out with a Smagorinsky constant of $C_s = 0.1$, which is a typical value for practical applications of Smagorinsky model.

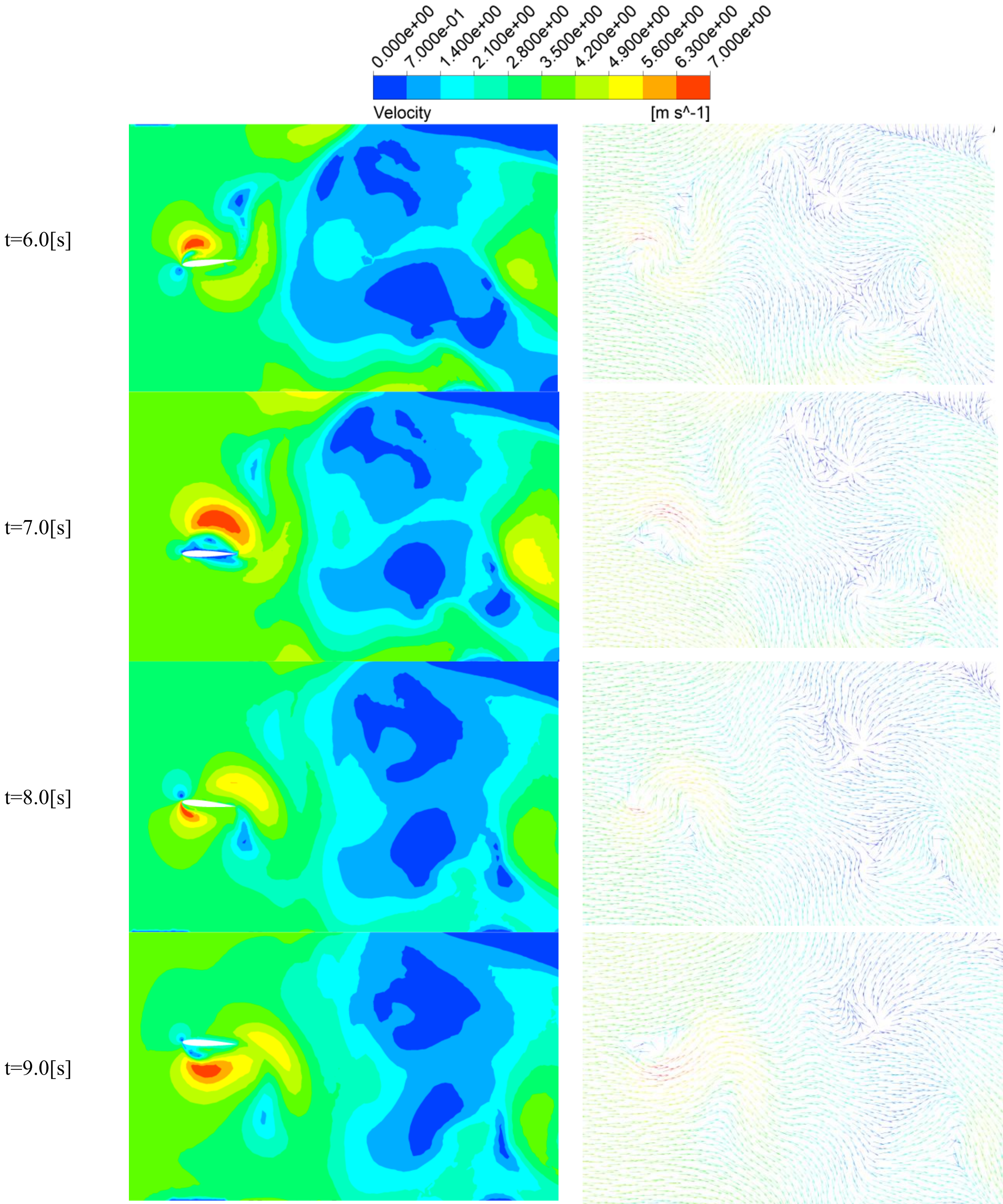
More complex SGS models have appeared in the literature (for example, the dynamic SGS eddy viscosity models), but they are beyond the scope of the present work.

III. Computational model

In the present research the subsonic flow past flapping finite wing NACA0012 is investigated using LES. The 3-D simulations were performed for Mach number for different flapping frequencies. The computational domain consists of 3.2 million grid points with a cluster of grid points around the wing and a grid expansion factor of 0.1 For all of the computations in this paper, a dimensionless time step $\overline{\Delta t} = \Delta t U_\infty / c = 7.5\text{x}10^{-6}$ is chosen, where U_∞ is the free- stream velocity. The time-step is determined with respect to the explicit time-marching scheme and temporal resolution requirement of LES (CFL $\leq$ 1). In the present investigations a value of Courant–Friedrichs–Lewy (CFL) number of 0.9 was chosen. The flow field is solved using the filtered Navier-Stokes equations along with a standard subgrid scale model and van Driest wall damping, $l = C_s \overline{\Delta} [1 - \exp(-y^+ / 25)^3]^{0.5}$.

IV. Results and Discussion

Figure 1 presents the velocity and pressure fields of the NACA0012 flapping wing. It is worth mentioning here that the wing undergoes a pitching and plunging motion following a sinusoidal path. The analysis of the flow field reveals that the flapping wing generates a highly turbulent wake and this best illustrated by the vector field. As the wing in accelerating in a plunging motion, high velocity magnitude is observed, above or underneath the wing depending on the direction of motion. The time-dependent velocity field influences the pressure field as well and this is shown in Figure 2.



a. Velocity magnitude b. Vector field

Figure 1. Flow field variables

Figure 2 presents the velocity magnitude and pressure field. From the velocity and pressure fields it is observed that the flapping motion, of the wing, has a significant effect on the flow field and it is expected to affect the aerodynamic coefficients, lift and drag, accordingly.

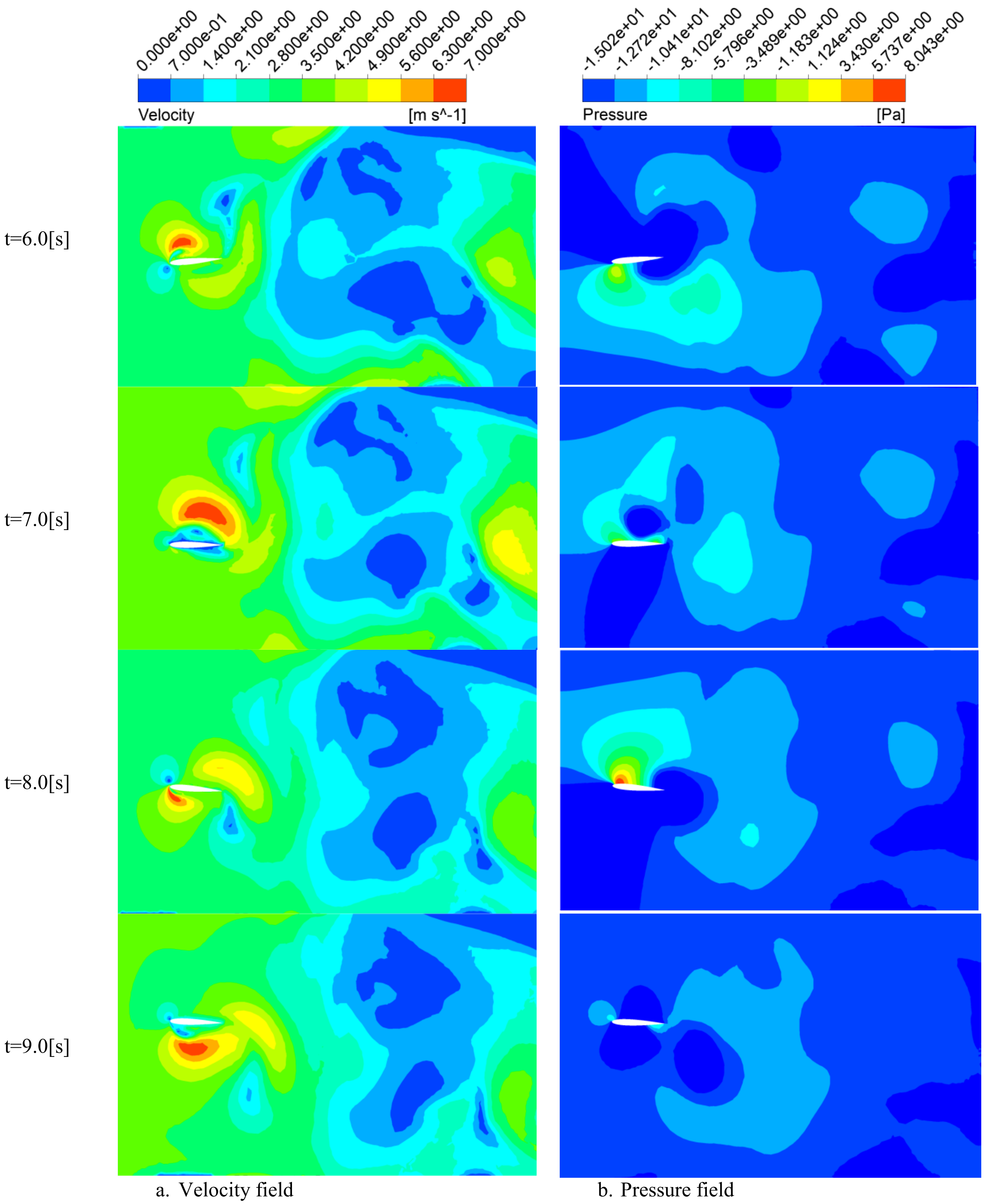


Figure 2. Flow field variables

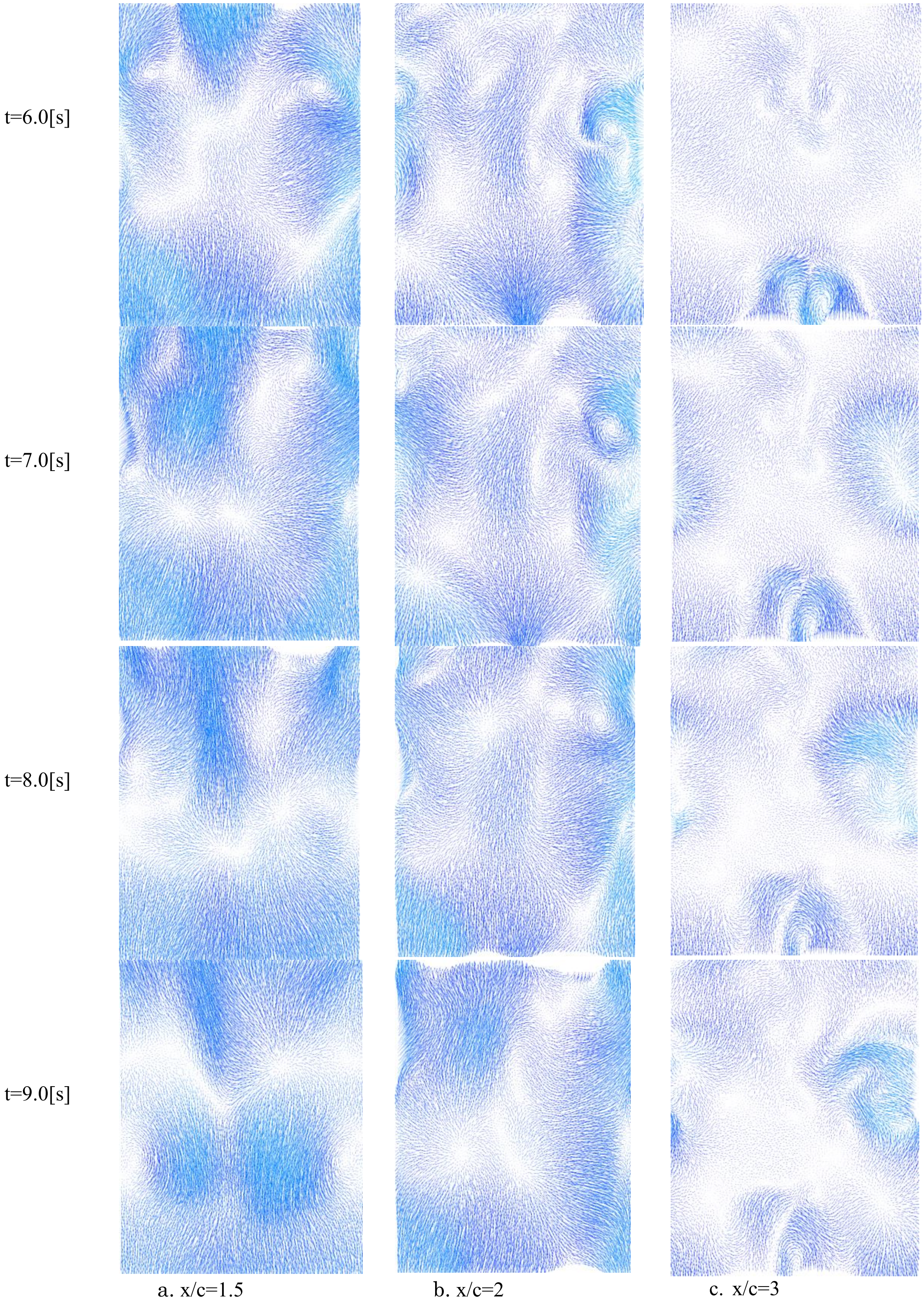


Figure 3. flow dynamics of the wake

Figure 3 presents the flow field in the wake of the wing, at different distances downstream. The analysis reveals the presence of a highly dynamic wake defined by large vertical structures whose intensity decreases with the stream wise distance, from the airfoil. However, the length scale of these vertical structures increase downstream the wing.

V. Conclusions

Subsonic flow past a flapping NACA0012 wing is investigated using LES approach. The analysis reveals that the presence of highly dynamic wake dominated by large scale and magnitude vertical structures. The flapping motion of the wing generates trailing-edge vortices whose magnitude and scale vary in time. The near-field flow is associated with periodic flow separation and reattachments, which cause a time-dependent aerodynamic coefficients. Although relatively expensive (from a computational point of view), LES can simulate fluid structures that cannot be captured by the unsteady Reynolds-Averaged Navier-Stokes, due to excessive numerical dissipation.

VI. References

[1] C. W. Shu and S. Osher, "Efficient implementation of essentially non-oscillatory shock capturing scheme," Journal of Computational Physics, vol. 77, 1988.

[2] S. Lele, "Compact finite difference schemes with spectral-like resolution," J. Comp. Phys., vol. 16, 1992.

[3] L. Jiang, H. Shan, and C. Liu, "Non-reflecting boundary conditions for dns in curvilinear coordinates," in Recent Advances in DNS and LES-Proceedings of the Second AFOSR International Conference on DNS/LES, Rutgers-The State University of New Jersey, New Brunswick, June 1999.

[4] H. Shan, L. Jiang, and C. Liu, "Direct numerical simulation of flow separation around a NACA 0012 airfoil," Journal of Computer and Fluids, 2005.

[5] P. Klebanoff, K. Tidstrom, and L. Sargent, "The three dimensional nature of boundary layer instability," J. Fluid Mech., vol. 12, 1962.

[6] T. Herbert, "Three-dimensional phenomena in the transitional flat-plate boundary layer," in AIAA paper, 1985-0489.

[7] C. F. Knapp and P. J. Roache, "A combined visual and hot-wire anemometer investigation of boundary-layer transition," AIAA Journal, vol. 6, 1968.

[8] K. Y. S. and L. V. Ya., "The resonant interaction of disturbances at laminar-turbulent transition in a boundary layer." J. Fluid Mech, vol. 138, 1962.

[9] R. D. Joslin, C. L. Streett, and C.-L. Chang, "Spatial direct numerical simulation of boundary-layer transition mechanisms: Validation of pse theory," Theoret. Comput. Fluid Dyn, vol. 4, 1993.

[10] L. Jiang, H. Shan, and C. Liu, "Direct numerical simulation of boundary-layer receptivity for subsonic flow around airfoil," in Recent Advances in DNS and LES Proceedings of the Second AFOSR International Conference on DNS/LES, Rutgers-The State University of New Jersey, New Brunswick, June 1999.

[11] D. Benney and C. C. Lin, "On the secondary motion induced by oscillations in a shear flow," Phys. Fluids, vol. 3, 1960.

[12] F. Ducros, P. Comte, and M. Lesieur, "Large-eddy simulation of transition to turbulence in a boundary layer developing spatially over a flat plate," J. Fluid Mech, vol. 1, 1996.

[13] M. El-hady and T. A. Zang, "Large-eddy simulation of nonlinear evolution and breakdown to turbulence in high-speed boundary layers," Theo. Comp. Fluid Dynamics, vol. 7, 1995.

[14] G. Erlebacher and M. Y. Hussaini, "Numerical experiments in supersonic boundary-layer stability," Theo. Comp. Fluid Dynamics, 1990.

[15] P. A. Elofsson and P. H. Alfredsson, "An experimental study of oblique transition in plane poiseuille flow," J. Fluid Mech, vol. 358, 1998.

[16] K. Y. S. and L. V. Ya., "Nonlinear development of a wave in a boundary layer," Fluid Dynamics, vol. 12, 1978.

[17] S. Berlin, M. Wiegel, and D. S. Henningson, "Numerical and experimental investigations of oblique boundary layer transition," J. Fluid Mech, vol. 393, 1999.

[18] P. A. Elofsson and P. H. Alfredsson, "An experimental study of oblique transition in a blasius boundary layer flow," Eur. J. Mech. B/Fluids, vol. 19, 2000.

[19]K. S. Breuer, J. Cohen, and J. H. Haritonidis, “The late stages of transition induced by a low-amplitude wave packet in a laminar boundary layer,” J. Fluid Mech, vol. 340, 1997.
[20]E. R. Van Driest, “Turbulent boundary layer in compressible fluids,” J. Aero Sci., vol. 18, 1951.
[21]M. Visbal and D. Gaitonde, “High-order accurate methods for unsteady vertical flows on curvilinear meshes,” in AIAA paper, Reno, Nevada, Jan 1998.

[22]D. Gaitonde and M. Visbal, “Further development of a navier-stokes solution procedure based on higher-order formulas,” in AIAA Paper 99-0557, Reno, Nevada, Jan 1999.
[23]T. Zang and M. Hussanini, “Multiple paths to subharmonic laminar breakdown in a boundary layer,” Phys. Rev. Lett, vol. 6, 1990.
[24]D. Anderson, Computational Fluid Mechanics and Heat Transfer. New York: McGraw-Hill Book Company, 1984.
[25]J. D. Anderson, Foundamentals of Aerodynamics. New York: McGraw-Hill Book Company, 1991.
[26]W. Zhao, Z. Liu, and C. Liu, “Direct numerical simulation of flow transition in a supersonic airfoil boundary layer,” in Proceedings of the First AFOSR International Conference on DNS/LES, Ruston, Louisiana, July 1997.
[27]E. Laurien and L. Kleiser, “Numerical simulation of boundary-layer transition and transition control,” J. Fluid Mech., vol. 199, 1989.
[28]C. Liu and Z. Liu, “Multigrid mapping and box relaxation for simulation of the whole process of flow transition in 3-d boundary layers.” J. Comp. Phys., vol. 119, 1995.
[29]H. Shan, L. Jiang, W. Zhao, and C. Liu, “Secondary instability of wall-bounded shear flows,” in AIAA paper, Reno, Nevada, Jan 1999.
[30]Z. Liu, G. Wang, and C. Liu, “Direct numerical simulation of flow transition in compressible swept-wing boundary layer,” in Proceedings of the First AFOSR International Conference on DNS/LES, Ruston, Louisiana, July 1997.
[31]H. Schlichting, Boundary Layer Theory. New York: McGraw-Hill Book Company, 1979.
[32]X. Zhong and M. Tatineni, “High-order non-uniform grid schemes for numerical simulation of hypersonic boundary-layer stability and transition,” Fluid Dynamics, vol. 190, 2003.