JOURNAL OF AIRCRAFT
Vol. 46, No. 5, September–October 2009

Evaluation of Computations and Transition Prediction Method for Aircraft High-Lift Configuration

Mitsuhiro Murayama,* Yuzuru Yokokawa,† and Kazuomi Yamamoto‡
Japan Aerospace Exploration Agency, Tokyo 182-8522, Japan
and
Yoshine Ueda§
Tokyo Business Service Company, Ltd., Tokyo 160-0023, Japan

DOI: 10.2514/1.34948

In this study, the three-dimensional flow computations over a realistic aircraft high-lift configuration with a flow-through nacelle with a pylon mounted beneath the main wing are performed using an unstructured mesh method to investigate the influence of the boundary-layer transition on the aerodynamic forces, the capability of a transition prediction method, and the influence of brackets to support the high-lift devices. First, the influence of the boundary-layer transition on the aerodynamic forces is shown by comparison of the computational results with/without the boundary-layer transition. Then a transition prediction method based on the e^N method and semi-empirical approaches is evaluated. The capability and areas to be improved are discussed. The influence of brackets to support the high-lift devices on the aerodynamic forces is also discussed. Interference of disturbed wakes by the slat supports to the flows on the main wing and flap is shown.

Nomenclature

C_D	=	drag coefficient
C_L	=	lift coefficient
$C_{L\max}$	=	maximum lift coefficient
C_M	=	pitching moment coefficient
C_p	=	surface pressure coefficient
c	=	mean aerodynamic chord
L/D	=	lift-to-drag ratio
M_∞	=	freestream Mach number
Re	=	Reynolds number
r	=	radius of curvature at the stagnation point
α	=	angle of attack
$\alpha_{\text{corrected}}$	=	angle of attack after wind-tunnel wall corrections
Λ	=	leading-edge sweep angle

I. Introduction

EFFICIENT high-lift devices of an aircraft for takeoff and landing can produce large benefits on payload and fuel consumption [1,2]. Recent computational fluid dynamics (CFD) technologies for three-dimensional configurations are expected to improve the performance. However, a multi-element high-lift wing system that uses leading-edge slats and trailing-edge flaps complicates the flow features, due to boundary-layer transition from laminar to turbulent states, flow separation, interactions of the wake of each element, and so on. For a realistic high-lift configuration with an engine-nacelle pylon mounted under the wing, the stall phenomena are often largely affected by the complex flow interaction between the nacelle pylon and the high-lift devices. The flows derived from the spanwise gap between the inner slat end and fuselage also often affect the stall performance. The precise prediction of the aerodynamic forces such as $C_{L\max}$ and L/D for the three-dimensional high-lift configurations is still a challenging task. The physical understanding of the dominant aerodynamic phenomena and efforts to improve CFD for such complicated flows over realistic high-lift configurations are required in conjunction with experiments [3–12] to improve the performance.

In recent years, efforts to validate and improve CFD for high-lift systems have been promoted [4–15]. In the European EUROLIFT [6–12] project, developments of aerodynamic analysis for high-lift configurations have been intensively conducted in conjunction with experiments. The Civil Transport Team of Japan Aerospace Exploration Agency (JAXA), has conducted a research program to develop design technologies for advanced high-lift devices. In the research, the first wind-tunnel testing using an aircraft configuration, denoted as the JAXA Standard Model (JSM), deploying the high-lift devices with fuselage, long-cowl flow-through nacelle pylon, and flap track fairings was conducted from October 2005–February 2006 to increase the knowledge of high-lift flows over a realistic aircraft configuration, to improve the measurement technologies, and to provide the detailed and systematic experimental data, which can be disclosed for CFD validation [16–19]. The first CFD workshop using the data was also held in the domestic communities in October 2006. In March–April 2007, the second wind-tunnel testing was conducted to obtain additional data for laminar–turbulent transition of the boundary layer, nacelle interferences, additional aerodynamic devices, and so on. The third wind-tunnel testing was conducted in December 2007.

The aerodynamic forces for low-speed high-lift configurations at the flight high-Reynolds-number condition are often extrapolated from the results at the subscale wind-tunnel Reynolds number conditions. The maximum lift is expected to increase with increasing Reynolds number. However, the adverse Reynolds number effects often occur in some cases. The change of laminar–turbulent transition of boundary layer by Reynolds number can be one of the reasons to cause the adverse Reynolds number effects [20]. In the prediction of aerodynamic forces for high-lift configurations, the effect of the transition to the aerodynamic performance should be well estimated with the transition prediction methods. The flow interferences due to the nacelle pylon mounted beneath the main wing and high-lift devices can cause the undesirable effects. Therefore, the evaluation in the complete aircraft configuration is also important.

Presented as Paper 3924 at the 25th AIAA Applied Aerodynamics Conference, Miami, FL, 25–28 June 2007; received 3 October 2007; revision received 2 May 2008; accepted for publication 5 May 2008.

*Researcher, Civil Transport Team, Aviation Program Group, 7-44-1 Jindaiji-Higashi, Chofu. Senior Member AIAA.

†Researcher, Civil Transport Team, Aviation Program Group, 7-44-1 Jindaiji-Higashi, Chofu. Member AIAA.

‡Section Leader, Civil Transport Team, Aviation Program Group, 7-44-1 Jindaiji-Higashi, Chofu. Member AIAA.

§Engineer, 6-14-1, NishiShinjuku, Shinjuku.