Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 99 (2015) 515 – 529

Procedia Engineering

www.elsevier.com/locate/procedia

"APISAT2014", 2014 Asia-Pacific International Symposium on Aerospace Technology, APISAT2014

# Investigation on Different Methods for Numerical Simulation of Propeller Slipstream

Zhang Ying, Ye Liang, Yang Shuo, Zhu Xinglin *

*Chinese Aerodynamic Research Institute on Aeronautics, Shenyang 110034，China*

**Abstract**

In order to describe propeller slipstream interactions with other components and assess the capability and applicability of different methods in propeller slipstream simulation, the flow fields and aerodynamics of propeller/nacelle and propeller/wing models at different work status are calculated. Numerical methods used for comparisons are unsteady method in which propeller is accurately simulated with body-fitted grid and quasi-steady approach where propeller is modeled as an actuator-disk. The solver used is based on unstructured embedded grid technique and OMP parallel method, Reynolds averaged Navier-Stokes equations associated with the one-equation Spalart-Allmaras turbulence model are solved and the governing equations are discretized by a second-order upwind finite-volume scheme and dual time-stepping scheme. The results for propeller/nacelle and propeller/wing configuration in axial flow conditions, which contain the induced velocity in axial and azimuthal directions and aerodynamics obtained by different methods, are compared. Meanwhile, effect of propeller installation to wing loads under different incidence flow angles is discussed further, and some meaningful conclusions are drawn.

Peer-review under responsibility of Chinese Society of Aeronautics and Astronautics (CSAA)

*Keywords:* propeller wing interactions; slipstreams; embedded grid; unsteady; actuator model; OMP parallel

## 1. Introduction

The interaction among propeller and other components brings an important influence to the aerodynamic characteristic of turbo-prop aircraft, which embodies in two aspects. One is the effects of propeller wake on other

* Corresponding author. Tel.: +86-24-86566610; fax: +86-24-86520827.
*E-mail address:* zyzq6583@126.com


Peer-review under responsibility of Chinese Society of Aeronautics and Astronautics (CSAA)
doi:10.1016/j.proeng.2014.12.566

components, which change the inflow magnitude, incidence angle and lift distribution of rear components. The other is the blockage effect, provided by wing or other asymmetric components, on propeller slipstreams, which will change the distribution of propeller thrust and torque. Therefore, predicting the interaction between propeller slipstreams and other components accurately and reasonably has been always an important issue during the aerodynamic study and design stage of turbo-prop aircraft, and a great number of researches have been conducted on it in recent years, see reference [1-3].

In the aspect of propeller slipstreams numerical study, developed methods can be classified as quasi-steady and unsteady with time terms discretization difference of governing equations. When propeller is simulated by actuator disk model[2,4], the real configuration of blades is not be described, which greatly reduces the amount of gird cells and decreases the difficulty of grid generation. Meanwhile, only additional source term needs to be added to the governing equations, which can be obtained by inquiring the airfoil Cl/Cd table or other ways, and the time consumption for calculation is significantly reduced due to the quasi-steady characteristic of flow fields. This method is a quasi-steady method in essence which is not suitable for 3D simulation, and the unsteady flow characteristics may not be estimated accurately due to the source term is obtained by 2D calculation.

In the unsteady method, the configuration of blades is accurately simulated and unsteady Navier-Stokes equations are solved in inertial reference frame. In order to conveniently describe the relative movement between propeller and other components, corresponding grid technique need to be used, such as slide grid technique[5], embedded grid technique[1,3,6] and etc. Similarities of these methods are that the grid system is multi-block, in which body-fitted grids around rotating propeller and unmoving components (such as wing or nacelle) are separately generated and data need to be exchanged among different grid blocks. The advantage of this method is that it is much more close to the real physical situation and can accurately describe the generation and variation of propeller wake with time. However, complicated grid system and a large number of grid cells will be needed when compared with actuator disk model and several propeller revolutions must be computed for obtaining periodical results, which bring the simulation more time and resource consuming.

The concerning contents are not the same at different stage of turbo-prop aircraft design or with different research work. For example, at preliminary design stage, a lot of aerodynamic estimations are required and interactions between propeller slipstreams and other components need to be rapidly appraised when propeller thrust is given in advance. At detailed design stage, more high-order numerical methods are needed to calculate propeller thrust, torque and figure of merit. The comparison of ability and applicability of different numerical methods in flow field details simulation and overall aerodynamics prediction can provide a reference for methods selection in turbo-prop aircraft design.

Flow fields and aerodynamics of propeller/nacelle model and propeller/wing model are simulated by quasi-steady and unsteady method separately, aiming at finding out the results discrepancy and giving the application range of different methods. The results which are used for comparison mainly include the induced velocity in axial and azimuthal directions after propeller disk, and aerodynamics distribution on wing at different angles of attack. Embedded grid system is used for simulation, in which minor grid and background grid are generated around propeller and other components. The same background grid is used in the above two methods so that grid discrepancy is only limited to a small region around propeller disk, which makes the results comparable.

Reynolds averaged Navier-Stokes equations are spatially discretized by an upwind finite-volume scheme and second-order precision is obtained by gradient reconstruction technique, the LU-SGS implicit scheme is used for time stepping and dual time-stepping scheme is taken into account for unsteady conditions, the solve process is accelerated by OMP parallel technique.

The comparisons show that discrepancy among lift distribution on the wing, axial induced velocity behind propeller disk obtained by the quasi-steady and unsteady method is not obvious when propeller thrust is given in advance, although the azimuthal induced velocity reflects intense unsteady characteristics. The existence of the propeller changes the magnitude and direction of inflow, which contribute to the change of the lift distribution on the wing, and these changes are related not only to the work status but also to the location of the propeller installation.