Ocean Engineering xxx (2017) 1–17

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Comparison of different approaches tracking a wing-tip vortex

D.-F. Feder [a,*], M. Dhone [b], N. Kornev [b], M. Abdel-Maksoud [a]

[a] Hamburg University of Technology (TUHH), Institute for Fluid Dynamics and Ship Theory, Am Schwarzenberg-Campus 4 (C), 21073 Hamburg, Germany
[b] University of Rostock, Chair of Modelling and Simulation, Albert-Einstein-Straße 2, 18059 Rostock, Germany

ARTICLE INFO

Keywords:
Tip vortex
Grid free vortex method
Turbulence modelling
Adaptive mesh refinement
Adaptive vorticity confinement

ABSTRACT

This paper compares the performance of different grid based and grid free modelling approaches to predict the tip vortex evolution in both near and far wing wake fields. The grid based methods cover different turbulence modelling approaches, adaptive mesh refinement and the adaptive vorticity confinement (VC) method using the OpenFOAM code. Computational vortex method (CVM) coupled with the OpenFOAM simulation of the near field is utilised to properly predict the tip vortex behaviour in the far field. All simulation results are compared to results of the wind tunnel experiments conducted by Devenport et al. (1996). The comparison is based on the analysis of the vortex core parameters: the core size, the peak tangential velocity and the axial velocity deficit. Additionally, the results are compared with another numerical study by Wells (2009, 2010). It turns out that turbulence modelling plays an important role since simple one and two-equation models overpredict the turbulence intensity in the vortex core resulting in its fast decay. The potential of the adaptive VC method depends on the underlying turbulence model. Grid free vortex method shows a good potential to improve the simulation accuracy.

1. Introduction

Wing-tip vortices decay slowly and extend far downstream in the wake. Strong tip vortices with a large lifetime can be observed in many engineering applications with (rotating) lifting surfaces. In marine engineering the most important application is the marine propeller hydrodynamics. In case of submerged ship propellers tip vortex cavitation may create noise and lead to erosion of the rudder. The noise prediction is of big importance because it disturbs animals like whales and reduces comfort of people on board ship. The propeller slipstream including the tip vortices prescribes the flow at the rudder and therewith influences the rudder forces. Lifting foils are used in shipbuilding to damp ship roll motions and for the creation of the dynamic lift on hydrofoil ships. In these cases the trailing tip vortices may influence the inflow of the propeller or foils moving downstream in the wake. A further important area where tip vortices play a dominant role is the wake of aircraft. The evolution of the tip vortices of landing and starting aircraft has been extensively studied with respect to the hazard of wake encounter by other craft flying close to the big one. Also for Wing-in-Ground (WIG) effect vehicles the wake and tip vortex evolution has a strong influence on the flight stability (see e.g. Kornev and Matveev (2003), Rozhdestvensky (2006)).

The prediction of tip vortices with computational fluid dynamics (CFD) is a challenge as inherent artificial dissipation effects lead to an unphysically strong decay of the vortices. The artificial dissipation or numerical diffusion results from numerical errors due to the discretisation and turbulence modelling. Feder and Abdel-Maksoud (2016b) provide further information on this topic in conjunction with the numerical prediction of tip vortices. An accurate numerical prediction of the tip vortex evolution is limited by a huge amount of necessary computational resources and imperfection of available mathematical models.

Several numerical and experimental studies deal with the evolution of tip vortices and the difficulties concerning their prediction. Gerz, Holzäpfel and Misaka worked extensively on the numerical prediction of tip vortices within the wake of aircraft, see e.g. Gerz et al. (2002), Holzäpfel et al. (2003), Misaka et al. (2013) and Stephan et al. (2014) or Chow et al. (1994). Their approach is usually based on the application of large eddy simulations (LES) targeting to predict the vortex decay of aircraft including the influence of the ground effect on the tip vortices and their evolution in the atmospheric boundary layer.

Several studies deal with the analysis of tip vortex evolution in the near field, so up to a few chord lengths downstream. An interesting study is published by Samal et al. (2013) who investigated the flow structure of a wing-tip vortex behind a sweptback and tapered NACA 0015 wing at

* Corresponding author.
E-mail addresses: dag.feder@tuhh.de (D.-F. Feder), mahesh.dhone2@uni-rostock.de (M. Dhone), nikolai.kornev@uni-rostock.de (N. Kornev), m.abdel-maksoud@tuhh.de (M. Abdel-Maksoud).

https://doi.org/10.1016/j.oceaneng.2017.09.036
Received 20 December 2016; Received in revised form 22 August 2017; Accepted 24 September 2017
Available online xxxx

$Re = 181000$. Their simulation results showed a good agreement with experimental results. Further studies by Nash'at et al. (2013) have been conducted to analyse the wake in the near field of a NACA 0012 wing. This paper focuses on grid based and grid free methods to capture the details of vortices especially further downstream after the vortex has rolled up and started to decay. Kornev and Abbas (2016) studied CFD performance to predict the near vortex field in the wake of an oscillating wing at distances of one and half chords from the trailing edge. They showed a good agreement with experimental results of Birch and Lee (2005).

Experimental investigations of wing-tip vortices in wind tunnels confirm that tip vortices extend for long distances in the wake. Devenport et al. (1996) carried out an extensive study of a tip vortex trailing from a NACA 0012 wing until 30 chord lengths downstream of the wing. They present experimental data of turbulence properties as well as the tangential and axial velocities in the vicinity of the vortex core. The quality of the experimental data is superior to many other investigations because they were corrected for the wandering of the tip vortex core caused by instability of wind tunnel flow and possible wing vibration. Wells published a numerical study for the Devenport test case and analysed the performance of different turbulence models (see Wells (2009); Wells et al. (2010)). In the near field, the accuracy of the simulation results using a Reynolds stress turbulence model is very good. Further downstream, Wells observes excessive diffusion of the tip vortex which is not supported by experimental results. Within this study, different grid based and grid free simulation approaches are validated for prediction of the tip vortex evolution using the Devenport test case.

According to the measurements of Devenport et al. (1996) the flow in the vicinity of the vortex core is laminar. Utilisation of standard Reynolds-averaged Navier-Stokes (RANS) turbulence models based on Boussinesq approach leads to an overprediction of the turbulence viscosity within the vortex core which in turn results in its increased decay rate. As a remedy of this disadvantage, several improved turbulence models, validated in this paper, were proposed by different authors: curvature corrections for RANS models, hybrid RANS-LES methods and Reynolds stress transport models. The artificial numerical viscosity can be reduced by adaptive mesh refinement (AMR) and the adaptive vorticity confinement (VC) methods which are also in the focus of this paper. The first approach refines the mesh in the vicinity of the vortex core and the second approach introduces a momentum source term which should counteract the numerical diffusion. The VC method was developed by Steinhoff and colleagues in 1992 (Steinhoff et al. (1992), Steinhoff and Underhill (1994) and Steinhoff et al. (2005)) and further developed by several authors. It overcomes the significant deficiency of the previous VC formulations as it lacks the necessity of a user-defined forcing coefficient. Recent applications of the VC method showed its potential for ship propeller hydrodynamics, see e.g. Zhang et al. (2014). Another work to be mentioned here is presented by O'Regan et al (see O'Regan et al. (2016)). who demonstrated that the potential of VC is larger in conjunction with LES turbulence modelling than with any URANS approach. This behaviour was observed also by Feder and Abdel-Maksoud in Feder and Abdel-Maksoud (2016a) for the adaptive VC method.

2. Test case

This study targets to validate CFD models using the benchmark test case thoroughly studied in wind tunnel measurements presented in Devenport et al. (1996). The basic setup of the experiment is shown in Fig. 1. During the experiment, the evolution of a tip vortex generated by a rectangular wing with the NACA 0012 profile was studied. The wing with a blunt tip has the following dimensions: a span of 0.879 m and a chord length of $c = 0.203$ m. The wind tunnel has a quadratic test section of 1.83 m width and 7.33 m length.

Devenport et al. provide the most extensive data for the Reynolds number $Re = 530000$ (based on the chord length) and for 5° angle of attack. There, the vortex is tracked downstream until 30 chord lengths behind the leading edge of the wing. Experimental data are provided for axial and tangential velocity profiles through the vortex core and for turbulence properties (e.g. the turbulent kinetic energy). The target of the simulation approaches used in this paper is to predict the evolution of tip vortices until large distances downstream. For this purpose the test case of Devenport et al. is more informative than other experiments at which the tip vortices are tracked only in the near wake of wings.

Besides, another advantage of this wind tunnel data is the proper correction for the vortex wandering motion. This slow side-to-side movement is usually observed for wind tunnel generated tip vortices. Without a proper correction of this effect, the experimental data would suggest an increased vortex decay. The way how the experimental data is corrected is presented in details in Devenport et al. (1996). Feder and Abdel-Maksoud (2016b) did not observe wandering within a similar previous numerical study. This observation was based on the Hexpress mesh with 6.0 M cells and the SA-DDES turbulence model. The solver settings were equal to the ones used for the transient simulations within this study. As no wandering was observed within the simulations, the corresponding correction is not necessary. The postprocessing of the simulation results will be presented in Section 4.1.

3. Numerical setup

The simulations are conducted with OpenFOAM (see Weller et al. (1998)) on grids with hexahedral cells generated with Hexpress and structured grids generated with ANSYS ICEM CFD. The coordinate system is set according to Devenport et al. (1996) and displayed in Fig. 2a. The origin is placed at the leading edge on the wing-tip. The x-axis points downstream and the y-axis points along the wing. The computational domain is shown in Fig. 2a. The inlet boundary is located at $x/c = -7.4$ whereas the outlet at $x/c = 46.7$. The boundaries with y- and z-normals denote the walls of the wind tunnel's test section. The size of the domain's cross section corresponds to the measuring section of the wind tunnel and the location of the wing in the cross section is identical to the experimental one.

3.1. Grids

3.1.1. Hexpress low Re number grid

The low Reynolds number mesh was generated with Hexpress version 5.1 (the turbulent boundary layer is well resolved up to the viscous sublayer). Fig. 2a shows the (medium) mesh at certain boundaries.

An advantage of the Hexpress mesh is the possibility to attain a homogeneous cell size with a desired refinement level in the vicinity of the vortex core. The cell size around the tip vortex in the measurement section ($x/c = [5, 30]$) is chosen to obtain about eight cells inside the mean measured core diameter. The total cell number is approximately 6.0 M. Since the first cell height is small 2×10^{-5} m which yields $Y^+ \approx 1$, no wall functions are applied within the simulation. A few cell (about 20) layers are used within the boundary layer.

This mesh was also used within the previous studies by Feder and

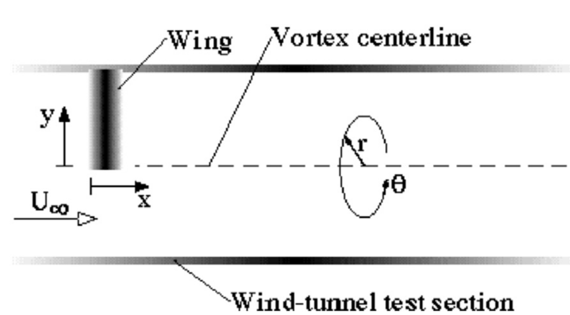

Fig. 1. Schematic view of the wind tunnel test section, from Devenport et al. (1996).