Implementation of the Spalart-Allmaras turbulence model in the two-dimensional vortex-in-cell method

Hejlesen, Mads Mølholm; Rasmussen, Johannes Tophøj; Larsen, Allan; Walther, Jens Honore

Published in:
Proceedings of the 6th European Congress on Computational Methods in Applied Sciences and Engineering

Publication date:
2012

Citation (APA):
Hejlesen, M. M., Rasmussen, J. T., Larsen, A., & Walther, J. H. (2012). Implementation of the Spalart-Allmaras turbulence model in the two-dimensional vortex-in-cell method. In J. Eberhardsteiner, H. J. Böhm, & F. G. Rammerstorfer (Eds.), *Proceedings of the 6th European Congress on Computational Methods in Applied Sciences and Engineering* Vienna University of Technology.

Implementation of the Spalart-Allmaras turbulence model in the two-dimensional vortex-in-cell method

Mads M. Hejlesen[†][*], Johannes T. Rasmussen[†], Allan Larsen[♯], Jens H. Walther[†][‡]

[†]Technical University of Denmark
Kgs. Lyngby, Denmark
s061937@student.dtu.dk

[♯]COWI Consulting Engineers A/S
Kgs. Lyngby, Denmark
ALN@cowi.dk

[‡]Computational Science and Engineering Laboratory, ETH
Zürich, Switzerland
walther@inf.ethz.ch

Keywords: CFD, unsteady flow computation, vortex-in-cell method, RANS turbulence modelling.

ABSTRACT

Most wind engineering problems deal with large bluff bodies that are located within the atmospheric boundary layer. For the cross section of large suspension bridges the relevant Reynolds number ranges from $O(10^5)$ to $O(10^8)$ with a typical turbulence intensity around 5-10% in the oncoming flow. In bridge aerodynamics, vortex methods are used extensively to calculate the aerodynamic forces and flow field around bridge cross sections due to its efficiency and inherent physical similarity, which is maintained even at large Reynolds numbers. So far the applied vortex methods have modelled the flow without employing an explicit model for unresolved turbulence scales. The hybrid vortex-in-cell (VIC) method offers a highly efficient particle-mesh algorithm that combines Lagrangian and Eulerian schemes to discretize different parts of the governing equation and thereby exploits the strength of each scheme. One of the benefits of the VIC method is its ability to efficiently solve transport equations. This is used in the present study to implement the transport based Spalart-Allmaras (SA) turbulence model [1] in the VIC method by an unsteady RANS approach. The two-dimensional VIC method, first presented in [2], uses a high order Poisson solver based on fast Fourier transforms subject to free space boundary conditions. By exploiting the linearity of the Poisson equation, high grid resolution is achieved using particle and mesh patches of different resolution. The final solution to the Poisson equation is given by the superposed solution of each patch subject to a free stream velocity U. Solid boundaries are implemented by a Brinkman penalization using an explicit split-step algorithm. The Spalart-Allmaras turbulence model is implemented by solving the eddy viscosity transport equation similarly to the vorticity equation by convecting the eddy viscosity along with the particles.

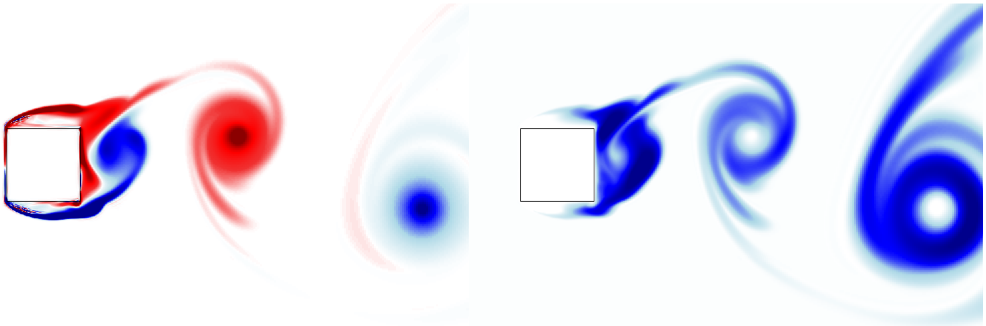

Figure 1: The simulated flow around a square cylinder using the Spalart-Allmaras turbulence model with an inflow condition of the eddy viscosity. Left: vorticity field. Right: modelled eddy viscosity.

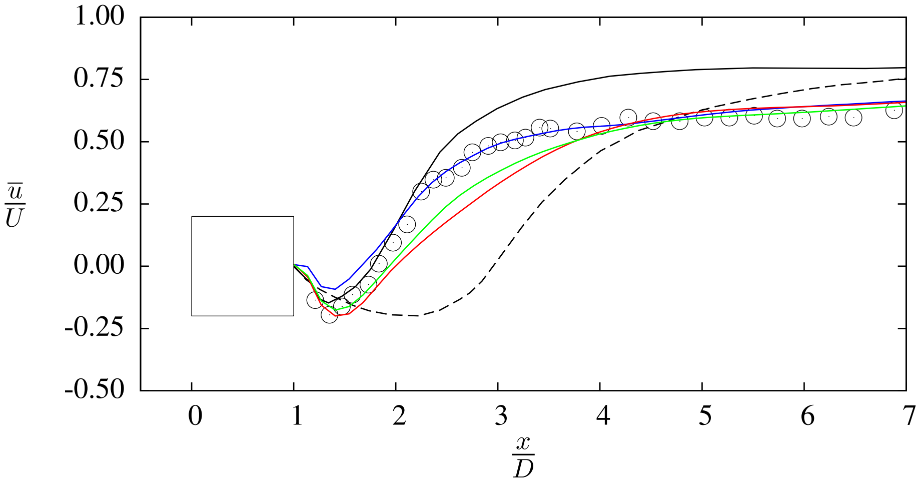

Figure 2: Wake profile of the time averaged horizontal velocity $\overline{u}$. Present results: ——: SA initial condition, ——: SA inflow condition, ——: SA trip term. Experimental results: ○ Lyn et al. [3]. Other 2D numerical results: —— and – – – –: RANS Bosch et al. [4]

Three different methods are used to assign an initial eddy viscosity to the particles: 1) by using the original trip term suggested by Spalart and Allmaras [1], 2) by using an initial field value that is convected out of the computational domain leaving only a self sustaining production of eddy viscosity, and 3) by defining an inflow condition of the eddy viscosity. As shown in figure 1, the implemented turbulence model is validated by simulating the highly separated flow around a square cylinder at Re $= UD/\nu = 22.000$, where D is the height/length of the cylinder and ν is the kinematic viscosity of the fluid. The resulting mean flow shows an overall good agreement with experimental data by Lyn et al. [3]. The method is able to capture the mean back-flow in the near wake of the cylinder as seen in figure 2. The aerodynamic drag coefficient is estimated to approximately 10% higher than the experimental data and the Strouhal number around 20% too high. The high Strouhal number is believed to be caused by the averaging of a general vortex shedding pattern resulting in a single vortex shedding instead of a double pairwise shedding, which is observed in experiments. Furthermore, preliminary investigations [5] on determining aerodynamic forces on bridge cross sections show promising results.

References

[1] P.R. Spalart, S.R. Allmaras: A one-equation turbulence model for aerodynamic flows. *Rech. Aerospatiale*, 1(1994), 5–21.

[2] J.T. Rasmussen, G.-H. Cottet, J.H. Walther: A multiresolution remeshed particle vortex method using patches. *J. Comput. Phys.*, 230(2011), 6742–6755.

[3] D.A. Lyn, S. Einav, W. Rodi, J.-H. Park: A laser-Doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder. *J.Fluid Mech.*, 304(1995),285–319.

[4] G. Bosch, W. Rodi: Simulation of vortex shedding past a square cylinder with different turbulence models. *Int. J. Numer. Meth. Fluids*, 28(1998),601–616.

[5] M.M. Hejlesen: Implementation of turbulence models in the vortex-in-cell method for simulation of bluff body aerodynamics. *M.Sc. thesis, Technical University of Denmark*, February 2012.