عنوان مقاله:
Computational Study of Flow around 2D and 3D Tandem Bluff Bodies

نویسندگان:

Terrance Charles - *School of Computing and Engineering, College of Science and Engineering, University of Derby, Derby, UK*

Zhiyin Yang - *School of Computing and Engineering, College of Science and Engineering, University of Derby, Derby, UK*

Yiling Lu - *School of Computing and Engineering, College of Science and Engineering, University of Derby, Derby, UK*

خلاصه مقاله:
Numerical simulations have been carried out to advance our current understanding of flow around two dimensional (2D) and three dimensional (3D) square shaped tandem bluff bodies at a Reynolds number of 22,000, especially to shed light on the sudden change of the downstream body's drag coefficient. The Reynolds-Averaged Navier-Stokes (RANS) approach has been employed in the present study and the predicted drag coefficients compare reasonably well with available experimental data. Better understanding of flow fields has been achieved by analyzing streamlines, velocity vectors for both 2D and 3D cases in a horizontal plane and a vertical symmetric plane. The sudden jump in drag coefficient of the downstream body for the 2D case is well captured numerically, which is due to the flow over the upstream body impinging onto the front face of the downstream body at a critical gap size between those two bodies. For the 3D case the drag coefficient is predicted to increase gradually, consistent with the previous experimental finding. This is due to the fact that the vortical structures formed in the 3D case are very different, resulting in a reasonably smooth change of the flow field around the upstream body and hence leading to a gradual, not sudden, increase in the drag coefficient of the downstream body.

کلمات کلیدی:
Tandem bluff bodies, Drag coefficient, RANS, Numerical simulation

لینک ثابت مقاله در پایگاه سیویلیکا:

https://civilica.com/doc/1222070