سیویلیکا - ناشر تخصصی مقالات کنفرانس ها و ژورنال ها
گواهی ثبت مقاله در سیویلیکا
CIVILICA.com

عنوان مقاله:
Numerical simulation of flow in open channel with a 90o bend

محل انتشار:
ششمین کنگره ملی مهندسی عمران (سال: 1390)

تعداد صفحات اصل مقاله: 8

نویسندگان:
S Baghalian - *M.Sc. student, Department of Civil Engineering, University of Razi, Kermanshah, Iran*

,H Bonakdari - *Water and Wastewater Research Center, University of Razi, Kermanshah, Iran*

M Fazli - *Assistant professor, Department of Civil Engineering, University of Bu-Ali Sina, Hamedan, Iran*

خلاصه مقاله:
A new methodology for resolve complex problems in fluid mechanics has been known as computational fluid dynamics (CFD). CFD investigates parameters such as velocity field, pressure, shear stresses, effect of turbulence and etc in flow field. In this paper, a three-dimensional CFD model was used to investigate of flow patterns, velocity profiles and also accuracy of numerical models in 1phase and 2 phases was investigated. For numerical simulation ANSYS-CFX software was used. The k-? turbulence model was used to solve turbulence equations. The results showed that secondary flow and centrifugal force influenced flow pattern and have good agreement with experimental

کلمات کلیدی:
numerical simulation, Computational Fluid Dynamic, bend, velocity, CFX

لینک ثابت مقاله در پایگاه سیویلیکا:
https://civilica.com/doc/120976