عنوان مقاله:

Numerical Modeling of the Stepped Planing Hull in Calm Water

محل انتشار:

ماهنامه بین المللی مهندسی, دوره 29, شماره 2 (سال: 1394)

تعداد صفحات اصل مقاله: 10

نویسندگان:

M Bakhtiari - *Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran*

S Veysi - *Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran*

H Ghassemi - *Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran*

خلاصه مقاله:

This article describes a 3D CFD (computational fluid dynamics) simulation implementation of the stepped planing hull in calm water. The turbulent free surface flow around the stepped planing hull is computed with a RANSE method, using the solver ANSYS-CFX. The turbulence model used is standard k–ε. In order to simulate the disturbed free surface, volume of fluid (VOF) model is implemented. The CFD model has been firstly validated using the available experimental data. The numerical results of drag, pressure distribution, wetted surface, water spray, wake profile and wave generated by the planing hull are presented and discussed at various speeds. Wake profiles calculated from present model are also compared with the ones calculated from Savitsky's emprical equations at different speeds.

کلمات کلیدی:

Stepped HullTurbulentFree Surface FlowVolume of Fluid ModelDragWake Profile