Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 194 (2017) 59 – 66

Procedia Engineering

www.elsevier.com/locate/procedia

10th International Conference on Marine Technology, MARTEC 2016

Numerical Prediction of Flow Past a Marine Rudder

Tahsinul Haque Tasif[a,*], Md. Habibur Rahman[a], Arafat Bin Fazle[a], Md. Mashud Karim[a]

[a]*Department of Naval Architecture and Marine Engineering, Bangladesh University of Engineering and Technology, Dhaka 1000, Bangladesh*

Abstract

In this study, flow past a rudder with NACA 0012 section has been analyzed using computational fluid dynamics (CFD). A commercial CFD code ANSYS Fluent based on Finite Volume Method (FVM) is used for analysis. At first, two-dimensional rudder section has been analyzed and computed results are validated by comparing with experimental results. Finally, flow past three dimensional rudder has been analyzed. For model development AutoCAD 14 and Rhinoceros 5 have been used. For mesh generation, ANSYS Workbench is used and for flow analysis Fluent is used. In case of two dimensional simulation, Transition k-kl-omega and SST k-omega turbulence models have been implemented to capture turbulent flow past the rudder. Again, in the case of three dimensional simulation, only Transition k-kl-omega turbulence model is used to capture turbulence. The simulation for both two dimensional and three dimensional rudders have been done at different angles of attack ranging from 0 to 20 degree. The computed results have been compared with the experimental results obtained by other researchers and found quite satisfactory.

Peer-review under responsibility of the organizing committee of the 10th International Conference on Marine Technology.

Keywords: Computational Fluid Dynamics, marine rudder; NACA 0012; turbulence model

1. Introduction

Rudder is the most important part of the steering arrangement in a ship or any other marine vehicle. It changes the course of the fluids past around the hull of the vessel and thus creates a yawing moment to the vessel. Using this yawing moment, the vessel changes its direction. When steering a vessel, a rudder creates both a lift force and a hydrodynamic drag force. If the lift force increases, the rudder will be more useful for turning but when the drag force increases, it increases the total resistance of the vessel. So it is very much important to choose a suitable rudder section so that, the lift force increases rapidly comparing to the drag force at different angels of attack. And for rudder it is expected that it produces similar force at different angles of attack for both port and starboard sides. In this study, NACA 0012 airfoil section is investigated to predict its suitability as a rudder section. NACA 0012 airfoil section is symmetrical about its chord length. So, it produces symmetrical lift and drag force for port and starboard turn. Here, the flow pattern of fluid around both 2D and 3D rudders has been studied using Computational Fluid Dynamics (CFD) with the help of Finite Volume Method (FVM).

This paper outlines numeric procedure to analyze the NACA 0012 airfoil section with a chord length of 1m and the Reynolds number 6×10^6 at different angles of attack from 0 degree to 20 degree with a step size of 2 degree.

* Corresponding author. Tel.:+8801675204035.
E-mail address: tahsinulhaquetasif@gmail.com

Peer-review under responsibility of the organizing committee of the 10th International Conference on Marine Technology.
doi:10.1016/j.proeng.2017.08.117

Nomenclature

c	Chord length
x	Position along the chord from 0 to c
y_t	Half thickness at a given value of x (centre line to surface)
t	Maximum thickness as a fraction of the chord
r	Leading edge radius
μ	Dynamic viscosity
ρ	Fluid density
ϕ	Flow properties
f	External forces
u_i	Velocity in X direction
u_j	Velocity in Y direction
u_{avg}	Mean flow velocity
u'	Root mean square of velocity fluctuations
I	Turbulence intensity
l	Length scale
L	Relevant dimension of the duct
k	Turbulent kinetic energy

2. Theoretical Background

2.1. NACA airfoils

The word NACA stands for National Advisory Committee for Aeronautics. The aerofoil shapes that are developed by NACA are mainly for aircraft wings. But these shapes are also used as aerofoils, hydrofoils, rudder sections and so on. Every NACA aerofoil has a distinctive name, which is a series of digits following the name "NACA'". The shape of the aerofoil depends on this series of digits.

In this study, NACA 0012 aerofoil section has been used, where the first two digits describe that there is no camber in the aerofoil and the last two digits describe that the maximum thickness of the aerofoil is 12% of the chord length.

2.2. Shapes for NACA airfoils

The thickness distribution for the NACA four digit sections is given by the following formula [1]:

$$\pm y_t = \frac{t}{0.20}\left[0.29090\sqrt{\frac{x}{c}} - 0.12600\left(\frac{x}{c}\right) - 0.35160\left(\frac{x}{c}\right)^2 + 0.28430\left(\frac{x}{c}\right)^3 - .10150\left(\frac{x}{c}\right)^4\right] \tag{1}$$

Where c is the chord length, x is the position along the chord length from 0 to c, y is the half thickness at a given value of x and t is the maximum thickness as a fraction of the chord. The leading edge radius is given by the following formula:

$$r = 1.1019t^2 \tag{2}$$

Where, r is the leading edge radius.

2.3. Computational Fluid Dynamics

In order to understand the physical phenomena in terms of hydrodynamic forces arisen by a rudder over which water flow crosses, researchers have used some methodologies to acquire and analyse data. There exist some approaches