Numerical Investigation of a Ship Propeller Jet using Computational Fluid Dynamic (CFD)

Yung-Jeh Chu, Wei-Haur Lam*, Cindy Soon, Long Chen

Department of Civil Engineering, Faculty of Engineering, University of Malaya, Kuala Lumpur, Malaysia.

*wlam@um.edu.my (Corresponding Author)

ABSTRACT: The hydrodynamics of a ship’s propeller jet is an initial input to investigate the ship propeller induced seabed scouring. The study investigates the velocity and the turbulence behaviours of a ship’s propeller jet using computational fluid dynamics (CFD). The CFD modelling includes propeller creation, grid generation, boundary condition setting and the suitable turbulence model selection. The influences of propeller geometry to the velocity distribution within the jet are also been investigated using computational fluid dynamics. The research is expected to provide a modelling technique which is able to be used to predict the velocity and turbulent behaviour of a ship’s propeller jet.

(Ship propeller jet, CFD, OpenFOAM, Fluent)

INTRODUCTION

The impact of ship’s propeller jet is one of the causes of seabed scouring. Numerous efforts have been done to preserve the ecosystems of the seabed but not much research has been done on the prevention of the seabed scouring. The propeller jet is one of the causes of seabed scouring [10]. Some scientists aware of the problem and proposed various theories and method to predict the degree of seabed scouring. A review of equations to predict the propeller wash is also done by some researchers [5]. Some researchers chose to use experimental method to model the propeller wash in a smaller scale [4]. CFD is one of the methods used by some researchers to predict the ship propeller wash [1, 6]. The ability to predict the velocity profile produced by a ship’s propeller jet will enable the prediction of sea bed scouring. Prevention can be done through reviewing the velocity profile in order to reduce the risk of propeller induced seabed scouring.

Several methods and theories of prediction on the velocity profile of a ship’s propeller jet are discussed in this paper. Nowadays, the CFD developments are getting better than past few years with the rising of the computational power available. Some of the commercial solver such as Fluent 6.1 offers free tutorial guide downloadable online [2]. The tutorial guides for the pre-processor, Gambit 2.2.30 are adequate to train a beginner to be a competent CFD modeller [3]. The open source software such as OpenFOAM also offers a compatible user guide which enables the beginners to learn the basic execution of the solvers and utilities available [7]. CFD modelling is used to predict the velocity profile of a ship’s propeller jet in this study.

The direction of this study is to simulate the propeller wash using commercial software. Fluent 6.1 is the solver while Gambit 2.2.30 is the pre-processor used in this study. Effort is given into this study in order to simulate the ship propeller wash by using an open source program, OpenFOAM 2.1.x. Comparisons and comments on the results analysed from the commercial software, Fluent 6.1 and the open source software OpenFOAM 2.1.x with the results from analytical calculations is expected to be done in this study.

METHODOLOGY

This paper will concentrate on modelling ship propeller jet with CFD. Two types of solver will be used which are the commercial CFD software named Fluent 6.1 equipped with a mesh generation software named, Gambit 2.2.30. On the other hand, a free solver named OpenFOAM 2.1.x which is run in a Linux operating system is used to simulate the same propeller case. The results of the velocity profile generated by both solvers will be compared to each other. The effort to determine their variation from the analytical results is done.