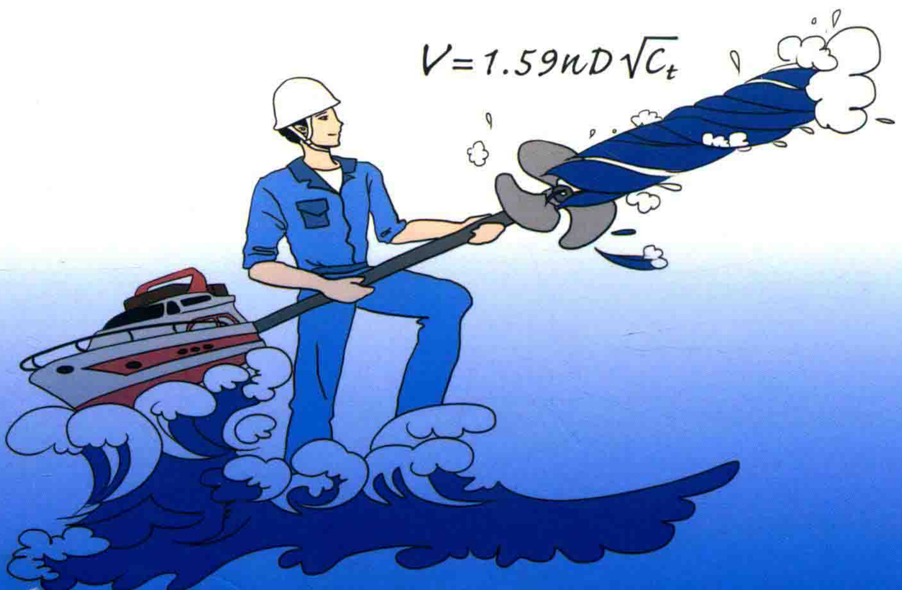
Introduction to Velocity and Turbulent Intensity within Ship Propeller Jet

船舶螺旋桨射流速度与湍流强度概论

林伟豪 | Wei-Haur Lam　著



天津大学出版社
TIANJIN UNIVERSITY PRESS

Introduction to Velocity and Turbulence Intensity within Ship Propeller Jet

船舶螺旋桨射流速度与湍流强度概论

林伟豪 | Wei-Haur Lam　著

天津大学出版社
TIANJIN UNIVERSITY PRESS

图书在版编目(CIP)数据

船舶螺旋桨射流速度与湍流强度概论：英文 /（马来）林伟豪著. — 天津：天津大学出版社，2017.1

ISBN 978-7-5618-5762-5

Ⅰ.①船… Ⅱ.①林… Ⅲ.①船用螺旋桨－尾流－船舶流体力学－研究－英文 Ⅳ.①U661.1

中国版本图书馆 CIP 数据核字(2017)第 033539 号

出版发行	天津大学出版社
地　　址	天津市卫津路 92 号天津大学内(邮编:300072)
电　　话	发行部:022-27403647
网　　址	publish. tju. edu. cn
印　　刷	廊坊市海涛印刷有限公司
经　　销	全国各地新华书店
开　　本	185mm × 260mm
印　　张	16. 5
字　　数	536 千
版　　次	2017 年 1 月第 1 版
印　　次	2017 年 1 月第 1 次
定　　价	55. 00 元

Abstract

This book is to investigate the velocity profile and turbulence intensity within a ship's propeller jet using a joint computational and experimental approach. A ship's propeller jet consists of two zones, the zone of flow establishment (ZFE) and the zone of established flow (ZEF). In these zones, the axial, tangential and radial components of velocity were predicted using computational fluid dynamics (CFD). The CFD predictions show the axial component of velocity is predominant in the propeller jet. This is followed by the tangential component of velocity, which contributes to rotation of the jet, and the radial component of velocity, which contributes to the jet diffusion. Laser Doppler Anemometry (LDA) was employed to measure the velocity magnitudes within these two zones allowing the validation of the CFD predictions. The investigation suggests the most appropriate choice of the turbulence model, rotating model, discretisation scheme, computational geometry and the grid type to be used to simulate a ship's propeller jet. Comparisons between the current experimental measurements and earlier works are also presented.

摘要

本书基于计算和实验两种方法研究船舶螺旋桨射流的速度分布和湍流强度。船舶螺旋桨射流包括两个区域：发展区（ZFE）和发展完成区（ZEF）。通过使用计算流体力学（CFD）方法，船舶螺旋桨射流的轴向、切向和径向的速度分量可以被预测，即螺旋桨射流中，轴向速度分量是最主要的分量；切向速度分量促使螺旋桨尾流发生旋转；径向分量最小，它导致尾流的发散。激光多普勒测速法（LDA）被用来测量这两个区域的速度大小，以此来验证 CFD 的预测结果。本书确定了适用于模拟船舶螺旋桨射流的湍流模型、旋转模型、离散格式、计算几何模型和网格类型。现有实验结果和前人工作的对比也在书中阐述。

Preface

Nowadays, many books focus on the structure and performance of the ship propeller, but books that introduce the flow structure within a ship's propeller jet are insufficient. This book introduces velocity distribution within the flow structure of a ship propeller jet by using Computational Fluid Dynamics (CFD) method and the experimental method.

The main contents of this book include: fundamental theory of ship's propeller jet, CFD simulation and the applications, LDA experimental works and turbulence intensity within a ship's propeller jet. The book introduces fundamental theories of plain water jet and ship's propeller jet based on the axial momentum theory. Readers are able to understand the development of the ship's propeller jet theory through pictures and theoretical derivation from axial momentum theory and plain water jet.

The book presents the applications of standard k-ε and standard k-ω turbulence models etc. to simulate the axial, tangential and radial components of velocity within the ship's propeller jet. Readers can understand CFD processes including the geometry creation, mesh generation and selections of the various physical settings to predict hydrodynamics of a rotating jet. LDA method is presented to measure the axial, tangential and radial components of velocity with comparisons of the CFD results. The book describes the LDA experimental setup, data acquisition and accuracy of the measurements. Readers can learn the LDA setup for experiments and also understand the flow structure within the jet with available information how to improve the jet theory through LDA results. This book is suitable for engineers, researchers, teachers, postgraduate and senior undergraduate students in the ocean engineering, harbour engineering, fluid mechanics, hydraulic disciplines.

Author thanks for the financial sponsor of researches from The Queen's University of Belfast under supervision of Dr. GA Hamill, Dr. DJ Robinson and Emeritus Professor S Raghunathan. The publication of this book was supported by the Science Fund for Creative Research Groups of the National Natural Science Foundation of China (Grant No. 51621092), Key State Laboratory of Hydraulics Engineering Simulation and Safety at Tianjin University, Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration at Shanghai Jiao Tong University, Key Laboratory of Ocean Energy Utilization and Energy Conservation of Ministry of Education at Dalian University of Technology and Institute of Ocean and Earth Sciences at University of Malaya.

前 言

目前研究船舶螺旋桨结构与性能的书籍有很多，而对船舶螺旋桨射流以及船舶螺旋桨射流对海底冲刷的研究则很少。本书系统地介绍了船舶螺旋桨射流的速度分布，并利用了计算流体力学方法和试验方法对其进行研究。

本书的主要内容包括：螺旋桨射流的基础理论、数值模拟方法的介绍与应用、激光多普勒测速法的试验研究、螺旋桨射流湍流强度的讨论。本书介绍了轴向动量理论、平射流理论等关于螺旋桨旋转射流的基础理论。读者可以通过图片结合理论方程对螺旋桨射流有直观的理解。

本书阐述了如何利用计算流体力学方法中的standard k-ε 和standard k-ω 等湍流模型对尾流进行模拟，预测径向速度，切向速度，轴向速度和湍流强度。读者可以了解如何建立螺旋桨几何模型、生成网格的方法，并学会了如何预测螺旋桨尾流的流场。试验利用激光多普勒测速法，对射流速度进行测量，并对实验布置、数据采集、误差来源进行详细的描述，与数值模拟的结论互为补充。读者可以从中学习如何设置激光多普勒测速仪，并深入了解螺旋桨射流的物理现象和理论模型的改进。本书适合从事海洋工程、港口工程、流体力学、水力学相关的工程师、研究员、教师、研究生、高年级本科生。

作者感谢英国贝尔法斯特女王大学对科研的资助，Hamill 博士、Robinson 博士和 Raghunathan 教授的在作者博士期间的细心指导。本书还获得国家自然科学基金创新群体项目(项目编号 51621092)、水利工程仿真与安全国家重点实验室、高新船舶与深海开发装备协同创新中心、教育部海洋能源利用与节能重点实验室、马来亚大学海洋与地球科学研究所的资助。

Author

Prof. Ir. Dr. Wei-Haur Lam is a full professor in ocean engineering, Tianjin University. He was formerly an associate professor in University of Malaya and visiting scholar in University of Oxford. He obtained his bachelor degree in civil engineering, master degree in computer science from University Technology Malaysia and his PhD from Queen's University Belfast. He is a Chartered Engineer from Engineering Council UK, Chartered Scientist from Science Council UK, professional member of Energy Institute, Chartered Energy Engineer, Chartered Engineer and professional member of Engineers Ireland, professional engineer from Board of Engineers Malaysia (BEM), professional member from The Institution of Engineers of Malaysia (IEM), member of International Association for Hydro – Environment Engineering and Research (IAHR), member of Malaysian Invention and Design Society (MIND), member of the Key State Laboratory of Hydraulics Engineering Simulation and Safety at Tianjin University and member of Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai Jiao Tong University.

作者简介

林伟豪,天津大学海洋工程专业教授、博导,曾是马来亚大学副教授、曾为牛津大学访问学者。他在马来西亚工艺大学获得土木工程荣誉学士学位、计算机硕士学位,在英国贝尔法斯特女王大学获得博士学位。他是英国注册工程师、英国注册科学家、英国能源学会专业会员、英国注册能源工程师、爱尔兰注册工程师、爱尔兰注册工程师学会专业会员、马来西亚注册工程师、马来西亚工程师学会会员、国际水环境工程与研究协会专业会员、马来西亚发明设计协会专业会员、天津大学水利工程仿真与安全国家重点实验室成员、上海交通大学高新船舶与深海开发装备协同创新中心成员。

Contents

目　录

Chapter 1 Introduction

The characteristics of the ship's propeller jet are firstly presented through the discussions of the axial, tangential and radial components of velocity and followed by the turbulence intensities. The applications of this complicated jet are also discussed throughout this book.

1.1 Characteristics of the Ship's Propeller Jet

The propeller wash is a complicated flow with axial, rotational/tangential and radial components of velocity. Experimental investigation is traditionally used to develop the predicting equations for the ship's propeller wash (Fuehrer & Römisch, 1977; Blaauw & van de Kaa, 1978; Berger *et al.*, 1981; Verhey, 1983; Hamill, 1987). However, the validity of these equations has not yet been confirmed for complex propeller geometries (van Blaaderen, 2006).

A rotating ship's propeller draws in water, accelerates and then discharges this water downstream to propel a ship. The discharge of water is a high velocity flow which is capable of scouring the bed if unchecked. In an unrestricted area, the velocity of the flow decays proportional to the distance from the propeller face by entraining the surrounding still water. If this jet is restricted, this high velocity jet will not decay naturally by entraining the surrounding water, but will cause damage to the adjacent area. If the movement of a jet is restricted by the sea bed, this will cause sea bed scouring, as documented by Hamill (1987). Hamill *et al.* (2009) included the effect of rudder angle on the velocity prediction of propeller wash.

The principle of propeller operation is to convert the torque of a shaft to produce axial thrust (Massey, 2005). The propeller provides this thrust by increasing the rearward momentum of the fluid in which it is submerged (Massey, 2005). As a reaction, the fluid exerts a forward force on the propeller which is used for propulsion. This fundamental concept provides the basis for all propeller propulsion theories (Stewart, 1992).

The flow field behind a manoeuvring ship, which encompasses a rotating propeller jet and turbulent wake from the hull, is complex in character. The rotating propeller produces a high velocity jet, while the bow cutting through the water induces a turbulent wake, making the flow pattern more complicated (Hamill, 1987). When the ship is stationary or manoeuvring at low speed, the wake due to the hull cutting through the water is insignificant and consequently the influence of hull's geometry on the propeller jet is negligible (Prosser, 1986). Approximation to the propeller wash can therefore be made with sole consideration of the action of the propeller (Prosser, 1986).

The velocity magnitude of a ship's propeller jet tends to decay along the longitudinal axis from the initial plane immediately downstream of the propeller jet (efflux plane) (Blaauw & van de Kaa, 1978; Hamill, 1987; Stewart, 1992; McGarvey, 1996). The eddies, which are generated in this region of high viscous shear, give rise to lateral mixing. As a consequence, the fluid within the propeller jet is gradually decelerated with longitudinal distance from the propeller face, while the still ambient fluid is gradually accelerated (Brewster, 1997).

The regions within a jet can be divided into two zones, the zone of flow establishment (ZFE) which lies close to the propeller face, followed by the zone of established flow (ZEF). As the propeller is rotating, the jet forms the zone of flow establishment initially. Due to the influence of the hub at the centre of propeller, the propeller jet has a low velocity core along the axis of rotation within the zone of flow establishment. The lateral velocity profiles within the zone of flow establishment are therefore two-peak ridges. The influence of the hub disappears gradually along longitudinal axis due to the penetration of high velocity fluid into low velocity central core (Hamill, 1987). At the beginning the fluid is mixing with the surrounding water both inwardly and outwardly along the axis of rotation (McGarvey, 1996). At a certain distance downstream, the flow will only be mixing outwardly and this is then referred to as the zone of established flow (McGarvey, 1996). In this region, there is only one maximum velocity peak located at the axis of rotation (McGarvey, 1996).

Researchers have found it difficult to develop equations for a wider range of propellers due to the expensive experimental setup required. The consideration of scale effects on a physical model also encourages researchers to use a computational approach to improve current equations used to predict the ship's propeller wash. The physical measurements of propeller wash are limited by the propeller size, with some propellers reaching 3m in diameter. Consequently, experiments of the propeller wash are normally undertaken with a scale model instead of a full scale propeller. The predictions made from the scale model experiment are limited by the scale effects.

Computational Fluid Dynamics (CFD) has been widely applied to investigate fluid dynamics problems and has become relatively inexpensive compared with experimental investigations. However, the application of CFD models is on a case by case basis. There is no universal CFD model that can be used in all the different cases (Fluent Manual, 2003). The aim of this research is to investigate CFD models, using the Fluent CFD package, to predict the velocity fields and turbulence intensities within the ship's propeller wash.

1.2 Applications of the Ship's Propeller Jet

The investigations into predicting the velocity within the ship's propeller wash which can lead to seabed scouring are of particular interest for the design of marine structures. In Whitehouse's (1998) book *Scour at Marine Structures*, the potential damage made by the propeller wash is