Ocean Engineering 87 (2014) 1–9

Contents lists available at ScienceDirect

Ocean Engineering

journal homepage: www.elsevier.com/locate/oceaneng

Unsteady RANS computations of flow around a circular cylinder for a wide range of Reynolds numbers

R.M. Stringer [a,*], J. Zang [a], A.J. Hillis [b]

[a] Department of Architecture and Civil Engineering, University of Bath, Bath BA2 7AY, UK
[b] Department of Mechanical Engineering, University of Bath, Bath BA2 7AY, UK

ARTICLE INFO

Article history:
Received 5 April 2013
Accepted 19 April 2014
Available online 11 June 2014

Keywords:
CFD
Cylinder
OpenFOAM
Tidal
URANS
VIV

ABSTRACT

A methodology for computation of flow around circular cylinders is developed and tested using prominent commercial and open-source solvers; ANSYS® CFX-13.0 and OpenFOAM® 1.7.1 respectively. A range of diameters and flow conditions are accounted for by generating solutions for flows at Reynolds numbers ranging from 40 to 10^6. To maintain practical solve times a 2D Unsteady Reynolds-Averaged Navier–Stokes (URANS) approach is taken. Furthermore, to maximise accuracy a tightly controlled meshing methodology, suitable adaptive timestepping, and appropriate turbulence modelling, are assembled. The resulting data is presented for lift and drag forces, Strouhal frequency, time accuracy and boundary layer correlation. Despite closely matching case definitions, significant differences are found in the results between solvers; OpenFOAM displays high correlation with experimental data at low to sub-critical values, whereas ANSYS proves to be more effective in the high sub-critical and critical regions. This variance demonstrates the sensitivity of the case to solver specific mathematical constraints and that for practical engineering a parameter study is essential. By removing many common variances associated with grid and transient components of URANS computations the developed methodology can be used as a benchmark case for further codes solving cylindrical structures.

1. Introduction

Understanding the flow around a circular cylinder has historically been a fundamental challenge for researchers, largely due to the complexity and transient nature of the wake. However, in the last decade, desktop computational resources have increased sufficiently such that high resolution solutions for practical engineering have become feasible. One such application, the motivation for the study, is the use of cylindrical geometries as structural members and pipelines in offshore applications. This usage is particularly relevant due to the exploitation of new renewable energy technologies both wind and marine, many of which include cylindrical features in some form. Analysis of circular cylinders for the offshore market has been primarily to assess structural loading caused by vortex shedding. This phenomenon has influenced new offshore technologies aimed at reducing the impact of vortex induced vibration (VIV) on structural elements such as riser fairings and platform leg surfacing. In the context of marine renewables, it is also possible that vortices shed from cylindrical components may reduce device efficiency and therefore require an increased level of resolution in design and development solutions. To address this, this research aims to develop and assess a rigorous numerical methodology for modelling such cases.

The flow around cylinders has been extensively investigated through experimentation by notable contributors such as Tritton (1959), Roshko (1955) and Achenbach (1968), amongst many others. One of the key outcomes of this work was to categorise flow by regimes of vortex shedding with Reynolds number (Re), given in Eq. (1). A prominent early paper by Lienhard (1966) proposes an outline of flow characteristics from laminar flow, up to supercritical values $\approx$ Re 3.5×10^6. However, the complexity of the turbulent wake has undergone many new discoveries, with a distinct contribution from advancing numerical modelling. A review by Williamson (1996) considers the wake in detail; highlights include a detailed account of the transition of wakes from 2D to 3D in the range $180 < Re < 190$, control of the shedding by modification of the cylinder end conditions, and the Direct Numerical Simulation (DNS) of 3D instabilities in landmark detail. The regimes of flow around a cylinder as Reynolds number increases have been refined by numerous researchers, most notably Zdravkovich (1990) with 15 distinct ranges. A summary of the key stages in flow development are presented in Table 1.

The study here considers incremental values of Re from 40 up to a maximum of 10^6. To give a perspective on the range, the peak

* Corresponding author.
E-mail address: r.m.stringer@bath.ac.uk (R.M. Stringer).

http://dx.doi.org/10.1016/j.oceaneng.2014.04.017
0029-8018/

Table 1
Flow regimes around a circular cylinder (Raghavan and Bernitsas, 2011).

Re range	Flow regime
$Re < 1$	Creeping flow
$3–5 < Re < 30–40$	Steady separation (Foppl vortices)
$30–40 < Re < 150–300$	Laminar periodic shedding
$150–200 < Re < 1.4 \times 10^5$	Subcritical
$1.4 \times 10^5 < Re < 1 \times 10^6$	Critical
$1 \times 10^6 < Re < 5 \times 10^6$	Supercritical
$5 \times 10^6 < Re < 8 \times 10^6$	Transcritical
$8 \times 10^6 < Re$	Postcritical

value of Re is equivalent to a 0.5 m pile in a 2 m/s tidal flow. This velocity range represents ‘slack water’ up to the peak flow/ebb for many locations around the UK, such as the Severn Estuary (Xia et al., 2010).

1.1. Numerical literature

At practical Reynolds numbers the wake and vortex formation around a circular cylinder is preclusively complex to fully compute, therefore a suitable level of spatial and temporal simplification has to be found. While it is known that 3D structures are common in the wake of circular cylinders, simplification of the case to 2D has been employed in the present research based on successful results obtained by a number of researchers.

Beginning with low Re cases, $Re < 160$, Park et al. (1998) and Dehkordi and Jafari (2009) both obtain excellent agreement with experimental values for all parameters monitored using a laminar URANS method; no ill effects from 2D simplification are found. Moving into the subcritical regime, research conducted by Rahman et al. (2008) compares a number of two-equation turbulence models at Re values of 1000 and 3900. Rahman et al.'s results show a clear improvement in accuracy using the shear stress transport model (SST) over the $k-\varepsilon$ and realizable $k-\varepsilon$ models. At critical and supercritical Re values of 10^6 and 3.6×10^6, Ong et al. (2009) evaluated the $k-\varepsilon$ model with a log law wall function. A limited study of the effect of y^+ was conducted although values are kept in the 5–30 region. Ong et al. compare their results with 2D and 3D Large Eddy Simulation (LES) and experimental data, with force and shedding frequencies falling within known limits. However, the pressure distribution and shear stress show some divergence. Benim et al. (2007) explored the topic of near wall meshing further by using the commercial code FLUENT to compute flow around a cylinder at $Re = 10^4$ using wall models and the standard $k-\varepsilon$ turbulence model. Benim et al.'s results from meshes in the range of y^+ values, from 10 to 1000, yielded a large range of drag values. Significantly no discernible plateau is visible. Consequently, the author continued testing without wall functions, switching to the SST turbulence model and adhering to meshes that conform to $y^+ = 1$. It is worth noting that a non-conformal surface grid is used, akin to a body fitted quadtree grid. In parallel with Ong et al., Benim et al. found acceptable correlation in the supercritical regime but this rapidly loses accuracy in the critical transition region, under-predicting values quantitatively for both $k-\varepsilon$ and, to a lesser extent, SST models. Tutar and Holdo (2001) computed results for the 2D case in both URANS and LES models at an Re of 1.4×10^5. Their results show that a non-linear two-equation $k-\varepsilon$ model gives improvement over the standard form, although both URANS methods under-predict pertinent values. LES is seen to produce a superior flow field, as expected, but results in over-prediction of force and shedding values compared with experiment. While LES in this case used a fully resolved boundary layer, the URANS method used wall models that have previously been shown to be highly mesh-dependent.

Based on the findings discussed here and additional sources, it can be concluded that the URANS method shows great promise for satisfactory prediction of flow characteristics around circular cylinders. However, the lines of applicability are blurred in terms of Reynolds number range and optimal computational methodology. This paper presents a rigorous methodology to overcome these limitations and to maximise the quality of URANS simulation. The methodology incorporates the SST turbulence model, a fully resolved boundary layer at every Re, a dense conformal grid, cell aspect ratio control and adaptive timestepping. Two solvers are used to compare the effects of the two host software packages, particularly as each uses alternative mathematical approaches.

The two software packages selected for the study are OpenFOAM 1.7.1 (OpenFOAM) and ANSYS® CFX-13.0 Academic Research (CFX). OpenFOAM is a C++ based open-source software written for the Linux platform, while CFX is a prominent commercial code heavily used in the aerospace and marine industries amongst others. Both OpenFOAM and CFX employ the finite volume method (FVM) to represent and solve the Navier–Stokes equations in algebraic form; Table 2 gives a basic outline of the contrasting approaches taken by the two solvers.

OpenFOAM has numerous FVM solvers depending on application, and for incompressible transient problems the pisoFoam solver is the most suitable. As the title suggests, pisoFoam uses the Pressure Implicit with Splitting of Operators method for pressure–velocity coupling proposed by Issa (1986). The method is akin to the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm with the addition of a second corrector stage that performs momentum (neighbor) and skewness correction. Note that all future references to OpenFOAM are specifically in regard to the implementation of the pisoFoam solver. As Table 1 states, OpenFOAM perfoms the PISO loop as part of the segregated solution method, while CFX uses a coupled solution, where continuity, momentum and energy are solved simultaneously and hence decoupling is avoided by using Rhie–Chow pressure interpolation. One of the key differences between the two methods is their sensitivity to timestepping. The coupled method in CFX is able to re-solve the governing equations in a pseudo inner timestep, whereas OpenFOAM converges each parameter once, correcting only for pressure and velocity in each timestep. The result is that CFX is relatively insensitive to timestep, while OpenFOAM requires tight control, such as adhering to low Courant numbers. In terms of spatial discretisation, the medium dual-method adopted by CFX divides the original mesh into a new set of polyhedral volumes defined by connecting the face centroids and edge midpoints of all cells that share any single grid node. In contrast, the cell centered method uses the existing cell volumes defined by the input mesh. The result is that CFX includes a greater number of integration points, while OpenFOAM retains a greater level of flexibility. A comprehensive comparison of the methods is offered by Blazek (2005).

Turbulence models are used equally for both solvers such that cases considered to have globally low Re employ a laminar model,

Table 2
Comparison of mathematical attributes for CFX and OpenFOAM.

Attribute	CFX	OpenFOAM[a]
Solution method	Fully coupled	Segregated
Temporal control	Implicit	Implicit/explicit
Discretisation	Median-dual cell-vertex	Cell-centred
Variable storage	Collocated	Collocated
Pressure–velocity handling	Rhie–Chow (adapted)	PISO

[a] Attributes specific to pisoFoam module.