CFD Simulations and Experimental Measurements of Flow Past Free-Surface Piercing, Finite Length Cylinders with Varying Aspect Ratios

M.A. Benitz^{a,*}, D.W. Carlson^b, B. Seyed-Aghazadeh^b, Y. Modarres-Sadeghi^b, M.A. Lackner^c,
D.P. Schmidt^a

Department of Mechanical and Industrial Engineering, 160 Governor's Drive, University of Massachusetts, Amherst, Massachusetts USA 01003

^a*Multiphase Flow Simulation Laboratory*

^b*Fluid-Structure Interactions Laboratory*

^c*Wind Energy Center*

Abstract

In this work, the flow past surface-piercing cylinders with free lower ends is studied numerically with computational fluid dynamics (CFD) at varying aspect ratios ($AR = H/D$) from around 1 to 19, and compared against experimental measurement across the same range. Numerical and experimental work is carried out at $Re = 2900$ and $Fr = 0.65$. The aspect ratio of the cylinder is varied by raising and lowering the cylinder into the water. The resulting flow field and loads on the cylinder as a result of changing the cylinder's depth of submergence are investigated in terms of drag, lift and frequency content. First, the effects of changing the aspect ratio are examined. CFD simulations demonstrate a reduced drag coefficient, consistent with the experimental measurements, as compared to a fully-submerged, infinite cylinder at the same Reynolds number. A significant reduction in the drag coefficient is observed at an aspect ratio of 2. For aspect ratios 3 and below, Karman vortex shedding is completely suppressed due to the dominance of free surface effects. With increasing aspect ratio the Strouhal number is found to approach 0.2. Second, the flow regime is investigated as a function of cylinder length for a cylinder with aspect ratio of 12. Different regions of vortex shedding are observed along the length of the cylinder, as demonstrated by lift forces, Strouhal numbers and CFD flow visualizations. Strouhal numbers below 0.2 were observed for regions near the free surface. Additionally, the drag coefficient was found to decrease near the free surface. This study over a wide range of aspect ratios exhibits the effects of a free surface, free end and also the length of the cylinder on both the flow regime and loading.

Keywords:

Surface-piercing cylinder, Finite cylinder, Drag coefficient, Vortex shedding, Volume of Fluid method

*Corresponding author

Email address: mbenitz@umass.edu (M.A. Benitz)

1. Introduction

Understanding the flow behavior around finite surface-piercing cylinders with small aspect ratios is of fundamental importance to many engineering applications, including floating offshore structures. Very few studies have addressed the combined factors of a free-surface and truncated end of a cylinder. Flow past infinite, fully submerged cylinders is well understood, due to ample experimental and numerical studies, including such works as Roshko (1961); Sabersky et al. (1998); Sarpkaya and Isaacson (1981); Schewe (1983) and Breuer (2000); Catalano et al. (2003); Grigoriadis et al. (2010); Kallinderis and Ahn (2005); Liang and Papadakis (2007); Lysenko et al. (2013); Perkovic et al. (2013); Sorenson et al. (2011); Travin et al. (1999); Wu and Young (2014); Zdrakovich (1997). Generous attention has been given to the isolated effects of finite cylinders, as well as surface-piercing cylinders. Far fewer research has been carried out, experimentally or numerically, on the simultaneous effects of a free end and free surface. Here, numerical investigations of surface-piercing, finite length cylinders are carried out over a wider range of aspect ratios than has previously been studied. The results are compared against experimental measurements, also performed by the authors. The effects of a free-surface and truncated end are discussed in terms of drag, lift and frequency content at Re and Fr numbers not studied previously.

Previous to this work, insight about the combined effect of these factors has been presented by a very small number of studies including Hay (1947), Goncalves et al. (2014) and Rosetti et al. (2013). The present work uses both experimental and numerical results to extend the range of studies to include a wider range of aspect ratios than has previously been presented, and also provides insight about the critical aspect ratio when a free surface is also present. Furthermore, the combination of the two factors is investigated, including which one, if any, is dominant across a range of aspect ratios. Additionally, sectional loads along the length of a cylinder with aspect ratio of 12 are given, providing insight about the flow behavior as it relates to a free surface and a free end. This investigation of loads as a function of length has not been previously presented for the case of a surface-piercing cylinder with a truncated end.

The numerical work was performed with unsteady Reynolds averaged Navier-Stokes (URANS) simulations, with a Spalart-Allmaras turbulence model. Experimental measurements were performed in a water tunnel, which is described in further detail below.

This work differs from previous work because it investigates the combined factors of a free-surface and free-end with numerical and experimental values across a wide range of aspect ratios not previously investigated. Furthermore, the present work provides loads as a function of cylinder length.

1.1. Surface-Piercing Cylinders

Surface-piercing cylinders have been investigated both experimentally and numerically. Experiments on surface-piercing cylinders were carried out by Akilli and Rockwell (2002) and Chaplin and Teigen (2003). Akilli and Rockwell (2002)’s experiments involved flow visualizations using visualization marker and particle image velocimetry. Chaplin and Teigen (2003) found that the local drag coefficient on the cylinder decreased near the still water line, in other words, near the free surface.

Numerical work on the drag coefficients of surface-piercing cylinders has been carried out by Kawamura et al. (2002), Yu et al. (2008) and Suh et al. (2011). In each of the papers, the drag coefficient for the cylinder in the presence of a free surface was found to be smaller than the subcritical constant value of 1.2 at Reynolds numbers below 1.4×10^5 for a fully submerged cylinder. This is consistent with the experimental findings from Chaplin and Teigen (2003). Previous work, both experimental and numerical, demonstrates that the presence of a free surface leads to a reduced drag coefficient.

In addition to effects on the drag behavior, the presence of a free surface has been shown to affect vortex shedding and lift forces. Yu et al. (2008) found that vortex shedding was suppressed near the free surface, which alters the lift forces acting along the length of the cylinder. Experimental work by Inoue et al. (1993) also found that vortex shedding was not seen near the free-surface.

1.2. Cylinders with Free Ends

Investigations of finite cylinders have been carried out both experimentally and numerically to investigate the effects of a free end on the overall flow regime. A thorough review of research on the flow field around free ends is given by Sumner (2013).

Experimental studies have shown the drag coefficient decreasing with reducing the aspect ratio. This includes work by Sumner et al. (2004), who investigated a vertically mounted cylinder on a flat plate. Farivar (1981) also studied drag coefficients of cylinders with small aspect ratios, and saw reductions in the drag coefficient. These relationships between drag coefficients and aspect ratio are consistent with other work in the literature by Sakamoto and Oiwake (1984), Taniguchi et al. (1981), Okamoto and Yagita (1973), Luo et al. (1996), Iungo et al. (2003) and Okamoto and Sunabashiri (1992). Reductions in the drag coefficient of finite length cylinders has also been demonstrated numerically, including work by . Lee et al. (2007), Pattenden et al. (2006) and Frohlich and Rodi (2004). Further numerical work on cylinders with free ends can also be found in Majumdar and Rodi (1989), Afgan et al. (2006), Palau-Salvador et al. (2010) and Krajnovic (2011).

Investigations of flow past finite cylinders of various aspect ratios has shown that the Karman vortex shedding is greatly affected by the free end of the body. Tip vortices that shed off of the truncated end of the cylinder interact with the Karman vortex shedding. Experimental measurements by Farivar

(1981) suggest that three different regions of shedding were present, indicated by different frequencies along the cylinder length. Kawamura et al. (1984) found that the Karman vortex shedding demonstrated dependence on the aspect ratio of the cylinder, such that at small enough aspect ratios periodic vortex shedding did not occur. The critical aspect ratio that marks the onset of Karman vortex shedding varies widely between authors, from 1 to 7 Kawamura et al. (1984); Farivar (1981); Okamoto and Sunabashiri (1992). These results are consistent with observations by Sakamoto and Arie (1983) and Park and Lee (2002) which showed reductions in Strouhal number near the free ends of the cylinders. Unique flow behavior has been noted in work by Park and Lee (2002); Adaramola et al. (2006); Rostamy et al. (2012); Roh and Park (2003); Pattenden et al. (2005) that investigated flow past small aspect ratio cylinders, which exhibited changes in the frequency content of the loads.

1.3. *Surface-Piercing Cylinders with Free Ends*

Investigations of the combined effects of the free-surface on finite cylinders has been given far less attention. Only a small sample of work in the literature has addressed the combined factors of a free surface and a free-end. In these studies, the work has excluded studying loads as a function of cylinder length. Except for the work by Hay (1947), the work has not examined aspect ratios larger than 2.

Hay (1947) conducted experiments in a towing tank and data were collected for the mean resistance and water run-up on the cylinder. The results showed peaks in the resistance coefficients at small Fr_L around 1, and then tapering of the resistant coefficient to nearly 0.5. This study is notable for its breadth of experimental parameters, it excluded discussion of frequency, vorticity and any measurements that vary with depth.

More recently, Rosetti et al. (2013), presented experimental and numerical investigations of a surface piercing cylinder at an aspect ratio equal to 2 at $Re = 4.3 \times 10^4$ and $Fr = 0.31$. It was found that the free surface affected the flow, but the free end effects dominated. Irregular vortex shedding was observed, as a result of the low aspect ratio. The global drag of the finite, surface-piercing cylinder was found to be smaller than the infinite, surface-piercing cylinder, and also smaller than the infinite, fully submerged cylinder. The observed drag and vortex shedding phenomena are consistent with other work in the literature. The study focused on global loads, excluding loads as a function of cylinder length. Furthermore, the study was limited to a single aspect ratio.

Goncalves et al. (2014), et al, presented thorough experimental results for varying aspect ratios up to 2 over a range of Re and Fr . The authors noted both a decrease in C_d and St with decreasing aspect ratio. In addition to load cell measurements, PIV measurements were also carried out. Visualizations of the flow around cylinders at a variety of aspect ratios were provided, showing the effects of a free end. The experimental data provided by Goncalves et al. (2014), complements the present work which presents numerical and experimental results up to $AR = 19$ with measurements at very small aspect

ratios. Furthermore, this present work adds to the existing data by providing results at different Re and Fr .

2. Analysis Specifications

The present work examines the combined effects of a free surface and a free-end on cylinders with varying aspect ratios. Both numerical and experimental work was carried out over a range of aspect ratios from around 1 to 19, where the aspect ratio is defined as $AR = \frac{\text{submerged length}}{\text{diameter}}$. The cylinder diameter was held constant at 0.0127 m and the depth of submergence was varied to alter the AR . A uniform flow of 0.23 m/s was propagated past the cylinder.

The numerical and experimental work is compared in an effort to further validate the numerical model as well as provide insight into the flow around surface-piercing cylinders with free ends and varying lengths. All of the results presented here are for the case of a $Re = 2900$ and $Fr = 0.65$.

Figure 1: Schematic of the experimental set-up for the surface-piercing cylinder in the FSI water tunnel. The depth of submergence of the cylinder can vary, allowing for the study of the relationship between aspect ratio and drag coefficient.

3. Experimental Method

The experiments were carried out in the re-circulating water tunnel of the Fluid-Structure Interaction Laboratory, with a test-section of $1.27\text{ m} \times 0.5\text{ m} \times 0.38\text{ m}$ and a turbulence intensity of less than 1% for up to a flow velocity of $U = 0.3\text{ m/s}$. A uniform circular cylinder with a diameter of $D = 1.27\text{ cm}$ was tested. Tests were conducted at a constant flow velocity of $U = 0.23\text{ m/s}$, resulting in a constant

Reynolds number of $Re = 2900$ (same as the Re number used in the numerical simulations). The Reynolds number chosen for this work is small compared to real world applications, due to limitations of water tunnel size and maximum flow rate of the water. The cylinder could be placed in water at different immersed lengths, resulting in different aspect ratios, L/D , where L is the immersed length of the cylinder (Figure 1). Twelve different aspect ratios of L/D between 0.875 and 19 were tested. A six-axis force sensor (ATI-Nano17/IP68) was attached to the upper end of the cylinder and measured the total flow forces acting on the cylinder in three perpendicular directions. This force sensor had a capability of high precision measurement of small forces. The calibration SI-25-0.25 had a sensing range of 25 N in the F_x , F_y and 35 N in F_z direction with a resolution of 1/160 N in all three directions. The single axis overload in F_{xy} and F_z were ± 250 N and ± 480 N , respectively. T_{xy} and T_z had a single axis overload capacity of ± 1.6 Nm and ± 1.8 Nm .

Four experimental runs were performed for each aspect ratio. The results presented in this paper include error bars illustrating $\pm$ one standard deviation from the mean of the four measurements taken at each of the aspect ratios.

4. Numerical Method

Numerical simulations were performed using OpenFOAM, an open-source finite volume continuum mechanics solver described by Weller et al. (1998).

4.1. Governing Equations

Numerical simulations were carried out using the incompressible Navier-Stokes equations, which are derived from the first principles of conservation of mass and momentum. The continuous partial differential equations are recast into a system of linear equations.

The Volume-of-Fluid (VOF) method was used to simulate the multiphase flow in this work, which consisted of air and water. In the VOF method the fluid phase is represented by a dimensionless scalar indicator function, α . The indicator function can be thought of as a fluid volume fraction, where a value of zero represents fluid “a,” and a value of one represents fluid “b,” such that the value of the indicator function represents the volume of a computational cell that is full of fluid “b.” The volume fraction is advected with the flow by a transport equation that is solved simultaneously with the equations of mass and momentum conservation. Cells with a value between zero and one represent the location of the fluid interface. Initial development of the VOF method can be credited to Noh and Woodward (1976), Nichols and Hirt (1975), DeBar (1974).

The indicator function, α , is given by Rusche (2002),

$$\alpha = \begin{cases} 0 & \text{for a point inside fluid "a"} \\ 0 < \alpha < 1 & \text{for a point inside transitional region} \\ 1 & \text{for a point inside fluid "b"}. \end{cases}$$

The indicator function, or fluid volume fraction, is advected with the flow, following a transport equation of the form,

$$\frac{\partial \alpha}{\partial t} + \mathbf{U} \cdot \nabla \alpha = 0. \quad (1)$$

Issues arise when advecting a step function such as the volume fraction, due to numerical diffusion. To mitigate these diffusion issues the interface can be compressed with an artificial compression term, U_r . This approach was first proposed by Weller, in Rusche (2002), also described in E. Berberovic et al. (2009), and is demonstrated mathematically below,

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\mathbf{U}\alpha) + \nabla \cdot (\mathbf{U}_r\alpha(1 - \alpha)) = 0. \quad (2)$$

4.2. Validation of the Numerical Method

The numerical model has been validated in multiple ways. Here, validations of the drag coefficient on infinite cylinders are presented across a range of Reynolds numbers. Furthermore, validation of wave run-up on a vertical cylinder is presented.

4.2.1. Drag coefficient validations

Prior to simulating flow past the surface-piercing cylinders with varying aspect ratios and free ends, validation of the numerical model was carried out on the simpler case of an infinite, fully-submerged, vertically mounted cylinder. The numerical drag predictions over a range of Reynolds numbers were validated against numerical and experimental values found in the literature.

A 0.0254 m diameter cylinder with a height of 0.1 m was placed in a 0.6 m by 0.6 m domain. Hexahedral meshes were generated, with the cylinder located at least 10 diameters from any of the domain boundaries. Octree refinement was employed to increase mesh refinement near the cylinder and fluid-interface region. No-slip boundaries were prescribed on the bottom and sides of the domain, and on the cylinder surface. A uniform velocity was prescribed on the inlet, which varied depending on the target Reynolds number of each simulation. Pressure was prescribed to have a normal gradient of zero on every boundary except for the top boundary, which was set with a total pressure boundary condition. The simulations presented here had an initially quiescent flow field (i.e. zero velocity everywhere in the domain). Adjustable time-stepping was employed, with a maximum Courant number of 0.5, where

$Co = U \frac{\partial t}{\partial x}$ and U is the velocity, and ∂t , ∂x are the discretized time and length intervals, respectively. The initial time step was set to 0.01 s.

The uniform flow velocity was altered to simulate the cylinder over a range of Reynolds numbers. At low Reynolds numbers, direct numerical simulation (DNS) was employed. At Reynolds numbers above 200 a turbulence model was used, as the meshing requirements for DNS at high Reynolds numbers become unrealistically large. Various turbulence models were tested, and are discussed in more detail below.

Figure 2: a) Force predictions from simulations with varying turbulence models, including Spalart-Allmaras, k-epsilon, k-epsilon RNG, and DES. Simulations with the Spalart-Allmaras turbulence model yielded the best results. b) Results from a sensitivity study of the turbulent viscosity initial conditions. It is shown that a $nu_t = 1 \times 10^{-6}$ gave the best results.

Turbulence models were methodically evaluated by extensive testing. Simulations were carried out with various turbulence models, including Spalart-Allmaras, k-epsilon, k-epsilon Re-Normalization Group (RNG) and Detached Eddy Simulation (DES), to determine which model produced the best results. The k-epsilon turbulence model was tested in some simulations, but the results exhibited excessive damping of the shedding vortices, as demonstrated by the lack of oscillation in the inline force signal shown in Figure 2a. The k-epsilon RNG model was also tested, and again, there was excessive damping of the vortices. At this Reynolds number, periodic behavior is expected due to vortex shedding, and that behavior was not captured with these turbulence models. Detached Eddy Simulations (DES) were also carried out, and the results underpredicted drag forces as compared to experimental data. It was found that the Spalart-Allmaras turbulence model resulted in the best agreement between numerical and experimental data. (The drag coefficients from simulations with the Spalart-Allmaras turbulence model are shown in Figure 4, and demonstrate good agreement with experimental data across a range of Re). Furthermore, as shown in Figure 2, oscillation of the loading due to vortex shedding was captured, unlike the results from k-epsilon and k-epsilon RNG. One might not expect the Spalart-Allmaras model, designed for slender airfoils, to be the choice of turbulence models. However, it outperformed the other

models.

Figure 3: An example of a mesh convergence study carried out at $Re = 20,000$. The cell count, normalized by the largest mesh's cell count, is shown on the x-axis and is plotted against the predicted drag coefficient. Experimentally determined drag coefficients are shown for comparison (Delany and Sorenson (1953); Roshko (1961); Sabersky et al. (1998); Sarpkaya and Isaacson (1981)).

Figure 4: a) The drag coefficient versus Reynolds number results across the full range of Reynolds numbers studied, b) the region of critical Reynolds numbers. Experimental work taken from the literature is shown in black lines (Achenbach (1968); Delany and Sorenson (1953); Fung (1960); James et al. (1980); Kovasznay (1949); Lindsey (1923); Relf and Simmons (1924); Roshko (1961); Sabersky et al. (1998); Sarpkaya (1977); Sarpkaya and Isaacson (1981); Schewe (1983); Triton (1959); Wieselsberger et al. (1923); Zdrakovich (1997)), and numerical work found in the literature is shown in blue circles (Breuer (2000); Catalano et al. (2003); Kallinderis and Ahn (2005); Perkovic et al. (2013); Singh and Mittal (2005); Song and Yuan (1990); Travin et al. (1999)).

Once the Spalart-Allmaras turbulence model was selected for simulations, a sensitivity study of the turbulent viscosity initial condition was performed. The results from this study are shown in Figure 2b. It was found that a turbulent viscosity value of $1 \times 10^{-6} Pa \cdot s$ for the initial condition yielded the best results. That is, the vortex shedding reached a quasi-steady state behavior, and the predicted mean inline force matched well with experimental results, as shown in Figure 4.

In addition to testing multiple turbulence models and carrying out a sensitivity study to the initial

conditions of the Spalart-Allmaras turbulence model, mesh convergence studies were also performed. Figure 3 shows an example of the solution convergence at $Re = 20,000$. A sample of experimental drag coefficients are plotted in the black lines (Delany and Sorenson (1953); Roshko (1961); Sabersky et al. (1998); Sarpkaya and Isaacson (1981)), and the drag coefficient predictions from CFD are shown at three different mesh sizes, increasing in cell count from left to right.

The results for the drag coefficient over a wide range of Reynolds numbers are presented in Figure 4a. It is shown that ample experimental (Achenbach (1968); Delany and Sorenson (1953); Fung (1960); James et al. (1980); Kovasznay (1949); Lindsey (1923); Relf and Simmons (1924); Roshko (1961); Sabersky et al. (1998); Sarpkaya (1977); Sarpkaya and Isaacson (1981); Schewe (1983); Triton (1959); Wieselsberger et al. (1923); Zdrakovich (1997)) and numerical (Breuer (2000); Catalano et al. (2003); Kallinderis and Ahn (2005); Perkovic et al. (2013); Singh and Mittal (2005); Song and Yuan (1990); Travin et al. (1999)) work has been performed to study the drag coefficient across a wide range of Reynolds numbers. The experimental and numerical predictions vary widely, especially in the region of higher Reynolds numbers.

An enlarged view of the high Reynolds number range is given in Figure 4b. The results from our work fall among the other authors' accuracy of prediction. These results demonstrate the ability of the CFD set-up to simulate uniform flow past stationary bodies, and to predict loads accurately.

4.3. Wave run-up validation

The model's ability to accurately capture the surface height of the water were validated through comparisons with experimental work by Niedzwecki and Duggal Niedzwecki and Duggal (1992). Comparison between experimental and numerical water height provides confidence in the ability to accurately model the air-water interface. The wave run-up is defined as the height of the water on the surface of the cylinder, measured from the still water level. The wave height follows the usual definition of trough to crest height. The amplification factor is calculated as the ratio of wave-run up to wave height, R/H . The numerical results were post-processed to determine the maximum fluid interface height on the surface of the cylinder, and the amplification factor was then calculated. The numerical and experimental results are shown in the left side of Figure 5. The experimental and numerical results show good agreement. Additional data for the wave height around the cylinder further demonstrates the ability of the model to capture the surface height of the water.

These additional data are presented as the water surface height around the cylinder as a function of theta, the angular location on the cylinder circumference. For the case where $ka = 0.175$, Niedzwecki and Duggal give the wave height for the run-up and run-down, where the wave is at its maximum and minimum, respectively, as a function of theta. Here, k is the wave number and a is the cylinder radius. Theta is zero at the front of the cylinder, and 180 at the back. Experimental measurements, results from diffraction theory and our numerical water height predictions from CFD are shown in Figure 5 on the

Figure 5: Amplification factor results for the stationary cylinder in regular waves. a) Wave run-up as a function of scatter parameter. The experimental results are shown in black stars, alongside the approximated fit shown in the dotted line, as well as the linear diffraction theory prediction. Numerical results from OpenFOAM as shown in red circles. b) Amplification factor around the cylinder as a function of theta, the angular location on the cylinder circumference. Numerical results are shown by the red and blue lines, indicating run-up and run-down, respectively.

right. The results show that OpenFOAM predicts the surface height around the cylinder for both wave run-up and run-down with good agreement to experimental results.

4.4. Numerical Set-up for Surface-Piercing Cylinders with Free Ends

Hexahedral meshes were generated for simulating the flow past the cylinders with varying aspect ratios. A mesh was built for each aspect ratio, where all meshes had a height of 0.55 m, width of 0.254 m and a length of 0.762 m. Each cylinder was located at 0.254 m from the inlet and 0.508 m from the outlet to allow room for the wake. The cylinder was located at a minimum of 10 diameters from any of the domain boundaries. Three layers of octree refinement were performed from the base mesh to the surface of the cylinder with an additional boundary layer on the surface of the cylinder. The boundary layer consisted of 10 layers. In addition to the increased refinement around the cylinders, the fluid interface region was refined with two layers of octree refinement in the vertical direction. The first layer of refinement fell within ± 0.025 m of the still water level (SWL), and the second layer of refinement fell within ± 0.0125 m. The cell count of each mesh varied slightly due to the differing lengths of the cylinders and the resulting numbers of refined cells near each cylinder surface. A mesh convergence study is presented for the case of $AR = 2$, in Figure 7, where the normalized cell count is plotted against the numerically predicted drag coefficient. The predicted drag coefficient is demonstrated to converge around 50% of the maximum cell count. The final mesh sizes used in this study were all in the neighborhood of 3.3 million cells.

The bottom and sides of the domain were treated as no-slip boundaries, as well as the surface of

Figure 6: a) View of the numerical domain from the side. Increased cell refinement is located around the fluid interface. b) Overhead view of the numerical domain. Increased cell refinement surrounds the cylinder. c) An enlarged view of the near cylinder region, as seen from above.

Figure 7: An example of a mesh convergence study carried out for the $AR = 2$ case. The cell count, normalized by the largest mesh's cell count, is shown on the x-axis and is plotted against the predicted drag coefficient.

the cylinder. The inlet was prescribed with uniform velocity of 0.2286 m/s in the water phase, and zero velocity for the air phase. The outlet boundary was prescribed with a zero-gradient for velocity. The volume fraction was given as 1 below the SWL and 0 above, where 1 represents water and 0 is air. Pressure was prescribed to have a normal gradient of zero on every boundary except for the top boundary, which was set with a total pressure boundary condition. The simulations presented here had an initially quiescent flow field (i.e. zero velocity everywhere in the domain).

Again, a maximum Courant number of 0.5 was set, with an initial time step of 0.01 s , and adjustable time-stepping.

The Spalart-Allmaras one-equation model, described in Spalart and Allmaras (1992), was used to model the turbulence, with a Spalding wall function at the cylinder boundary, presented in Spalding (1961). A y^+ value of 88 was used on the surface of the cylinder. The turbulence model was chosen

based on the findings from the validation exercises discussed in the validation work presented above.

An implicit Euler first-order time integration scheme was used, with adjustable time-stepping based on a maximum Courant number. A total variation diminishing (TVD) was used for the convection terms, which is a blending of upwind and central differencing, which is 1st/2nd order accurate. A Gauss linear gradient scheme, a linear central-differencing interpolation scheme and second order Gauss linear Laplacian scheme are used for the other discretizations.

Load predictions on the cylinder were calculated by discrete integration of the pressure and viscous stresses along the body, by summing values on each cell face. Moments were calculated in a similar manner. The individual moments from pressure and viscous forces on each cell face, where the moment arm was calculated as the distance from the face center to the cylinder center of gravity, were summed to yield a combined pressure and viscous moment on the body. The total moment is the sum of the pressure and viscous moments. The forces and moments were stored as three-component vectors in the x, y and z directions, and roll, pitch and yaw directions, respectively.

5. Results and Discussion

The results from CFD simulations, and the comparison with experimental measurements, are presented here. First, the drag forces predicted from simulations and measured in experiments are presented. Next, the frequency content of the load predictions from CFD is examined. Flow field visualizations from CFD are presented to provide insight about the flow physics around these surface-piercing, finite length cylinders of varying aspect ratios. Additionally, CFD results for sectional loads along a cylinder with an aspect ratio of twelve are presented. Finally, the section concludes with a comparison of the present results with previous work from the literature for the isolated cases of a free surface or a free end, as well as the limited cases where the two factors were studied together.

5.1. Drag Forces

The non-linear relationship between the drag coefficient and the aspect ratio is shown in Figure 8. The agreement between the numerical predictions and the measured data from the experiments are shown across the entire range of aspect ratios studied.

The drag coefficient is given by,

$$C_d = \frac{2F_d}{\rho U^2 A}, \quad (3)$$

where F_d is the time averaged drag force, ρ is the density of water, U is the freestream velocity, and A is the frontal projected area of the cylinder, which changes with aspect ratio. The numerical and experimental results show the same trend in drag coefficient versus aspect ratio. The drag coefficient is roughly constant for aspect ratios of 3 and above. A large drop in the drag coefficient appears at

an aspect ratio of 2, in both the experimental and numerical work. The reason for the drop in drag coefficient at $AR = 2$ will be investigated further through computational flow visualizations in a later section.

Figure 8: The drag coefficient at different aspect ratios. The numerical results are shown in red stars, and the experimental results are given in blue circles. A large decrease in the drag coefficient was observed both numerically and experimental for aspect ratios below 2.

Figure 9: The relationship between drag coefficient and aspect ratio from the present work as well as previous work from the literature (Farivar (1981); Okamoto and Yagita (1973); Okamoto and Sunabashiri (1992); Sakamoto and Oiwake (1984); Sumner et al. (2004); Taniguchi et al. (1981)).

To further investigate the combined effects of the free end and free surface on the drag behavior of cylinders, the relationship between the aspect ratio and drag coefficient is examined. In Figure 9 the present work is compared against results from previous work, where the cylinders had free ends, but did not pierce a free surface. (Experimentally, this can be achieved by mounting the cylinder vertically to

the floor, such that the free end is located at the top.)

Sumner et al. (2004) presents results that show initial signs of C_d plateauing between $AR = 5$ and 10, while the results by Sakamoto and Oiwake (1984), Okamoto and Yagita (1973), Okamoto and Sunabashiri (1992) and Taniguchi et al. (1981) fall in too narrow of a range of AR s to capture the plateau effect. The present work shows agreement with the general trends of lower drag coefficients at low aspect ratios, but also demonstrates a plateau at high aspect ratios. The differences between authors is most likely due to the different Re numbers at which these studies were performed. Overall, this comparison demonstrates that drag coefficients smaller than values for infinite cylinders at the same Re numbers are found for cylinders with free ends, excepting some results from Okamoto and Yagita (1973).

The discrepancies between previous work in the literature and the present work is likely due to the fact that the previous work investigated vertically mounted cylinders on flat plates, such that the free end was located at the top of the cylinder. In these works, the cylinders were partially immersed in a boundary layer attached to the floor, or flat plate. In the present work, the hard surface of the floor is replaced with a deformable water-air fluid interface.

Figure 10: Comparison of the numerically and experimentally determined drag coefficients from the present work, against numerical results by Rosetti et al. (2013) and experimental work by Goncalves et al. (2014).

Finally, experimental and numerical results from the present work are compared against numerical results by Rosetti et al. (2013) and experimental work by Goncalves et al. (2014), shown in Figure 10. The differences between work by Goncalves et al. (2014) and the present study indicate that load results are not universal, but rather that they depend greatly on the Fr and Re numbers. Future work will seek to understand this discrepancy in results, whether the difference is due to the Fr or Re , or a combination of the two.

5.2. Frequency Content

Fast Fourier transforms (FFTs) of the transverse force time histories from CFD were computed. The resulting FFTs are presented as a function of the Strouhal number, $St = fD/U$, where f is the vortex shedding frequency, D is the cylinder diameter and U is the uniform flow velocity. For a cylinder with $Re = 2900$, the St is expected to be 0.2 (Achenbach and Heinecke (1981)). The FFTs shown in Figure 11a show very strong peaks around $St = 0.2$ for the high aspect ratio cases of $AR = 15$ and 19. A less dominant peak is shown at $St = 0.18$ for the $AR = 9$.

Figure 11: FFTs of the transverse force signal for cylinders with aspect ratios of 3, 9, 15 and 19. Stronger peaks in the transverse forces are seen for cylinder with large aspect ratios. This indicates a large degree of periodic vortex shedding occurs in the wake of the longer cylinders.

An enlarged view of the FFTs is given in Figure 11b. At an aspect ratio of 6 a small peak is shown at $St = 0.15$, which is a slight reduction from the St numbers for larger aspect ratios. For aspect ratios 4 and 5, peaks are found at very low St . Finally, for $AR = 2$ and $AR = 3$ there exist no peaks in the FFTs, suggesting the absence of a strong oscillatory transverse force signal, and thus a lack of regular vortex shedding at these low aspect ratios. This analysis of the frequency content of the transverse forces demonstrates that the vortex shedding begins to be suppressed at an aspect ratio of 6, as the St decreases from the theoretical value of 0.2 for an infinite cylinder. The vortex shedding is increasingly suppressed as the aspect ratio is reduced, such that at an aspect ratio of 3 there exist no dominant frequencies in the vortex shedding.

Kawamura et al. (1984) noted the absence of Karman vortex shedding at $AR = 4$, while Okamoto and Sunabashiri (1992) noted suppression of vortices at $AR = 2$, which are roughly consistent with the present work's finding. The present work finds that Karman vortex shedding is suppressed at $AR = 3$ and below. Additionally, Sumner et al. (2004) found a reduced Strouhal number of 0.16 for cylinders with aspect ratios of 3, 5, 7 and 9 as compared to the value of 0.2 for infinite cylinders. The present work also demonstrates a reduced Strouhal number at $AR = 9$. However, at $AR = 3$ and below, periodic vortex shedding was not observed. This difference is likely due to the presence of a free surface which

did not exist in Sumner's study.

5.3. Visualizing the Flow

Here, flow field visualizations are presented in an effort to understand the underlying physics of the flow around surface-piercing, finite cylinders with varying aspect ratios. The previous subsections demonstrated non-linear relationships between the aspect ratio and the drag coefficients. Furthermore, the frequency content of the loads showed a dependency on the aspect ratio. Visualizations of streamlines and vorticity fields around the cylinders of various aspect ratios are presented here, and provide insight about the reasons for the load characteristics seen in the numerical predictions and experimental measurements.

Figure 12: View from above of streamlines, in 3d, moving from left to right past four cylinders of varying aspect ratio, including $AR = 2, 3, 9$ and 19 . The streamlines illustrate the increase in periodic vortex shedding with larger aspect ratios.

Views of the streamlines around cylinders with aspect ratios of $2, 3, 9$, and 12 are shown in Figure 12. These visualizations show overlapping streamlines located from just below the free surface to the bottom of the domain, as you move into the page. The cylinders with an aspect ratio of 2 and 3 do not appear to generate any vortices along the length of the cylinder. However, vortices are still generated from the free end. At an aspect ratio of 9 , vortex shedding occurs. The magnitude of the oscillation in the wake increases with increasing aspect ratio.

The streamlines are shown from the side in Figure 13. From this perspective, the absence of vortex shedding along the length of the cylinder in the $AR = 2$ and 3 cases is illustrated again. With increasing aspect ratio the vortex shedding again increases in magnitude. Near the free surface there is a lack of vortices in all cases, suggesting the free surface suppresses vortex shedding altogether. The ability of the free surface to deform, as opposed to a solid wall, alters the flow in this region. Surface waves are observed in both the experimental and numerical work. Farther away from the fluid interface, and

Figure 13: Streamlines past cylinders with four different aspect ratios, view from the side. The flow travels from left to right. Again, the increase in periodic vortex shedding is demonstrated as a function of the longer cylinder depth.

moving towards the free-end of the cylinder, the vortex shedding increases, such that the $AR = 19$ case exhibits vortex shedding clearly.

The side view of the streamlines also shows the portion of the flow that travels under the free-end of the cylinder and then moves upwards on the downstream side. The ability of the flow to travel beneath the cylinder because of the free end, as opposed to solely around the cylinder for the case of an infinite cylinder, is likely the main contributor to the reduced drag coefficient of finite cylinders.

Figure 14: Vertical slices through the numerical domain illustrating the vorticity fields around cylinders of aspect ratios 2, 3, 9 and 19, flow travels from left to right. Coherent periodic vortex shedding is observed in the vorticity slices of the cases with larger aspect ratios. The smaller aspect ratios show a smeared vorticity field which lacks periodicity.

The vorticity was computed from the CFD results. Vertical slices taken through the cylinder and parallel to the fluid flow are shown in Figure 14. At the small aspect ratio of 3, the vortex structure appears to be smeared out compared to the vortex structures behind cylinders with larger aspect ratios.

For the case of the large aspect ratio of 19, shown on the right of Figure 14, different regions of vortex shedding appear. Near the free surface the vorticity is very small. Along the upper-middle segment of the cylinder the vorticity oscillates periodically and parallel to the cylinder. Moving down to the lower-middle section, the vorticity remains oscillatory but the variation no longer occurs parallel to the cylinder, but rather at an angle. Finally, at the free-end of the cylinder there exists a different vortex structure altogether which is small and creates its own wake pattern. The tip vortices appear to interact with the Karman vortex shedding that occurs along the middle span of the cylinder. This effect is manifested in angled vortex sheets towards the bottom-middle section, while the top-middle section shows perfectly vertical vortex sheets.

Figure 15: Horizontal slices through the numerical domain of a cylinder with aspect ratio 9. The slices are taken at 10, 30, 50, 70 and 90% of the cylinder's depth. The patterns in the vortices in the downstream wake change as a function of the distance from the free-surface and free-end.

Horizontal slices of the vorticity field around a cylinder with $AR = 9$ are shown in Figure 15, at increasing depths below the still water level (SWL). The slices are taken at 10, 30, 50, 70 and 90% of the cylinder's depth, from top to bottom in the figure. The vortex shedding patterns differ along the length of the submerged cylinder. Near the free surface, shown in the top image, two vortices trail behind the cylinder and create a narrow wake. Moving down the length of the cylinder into deeper water, the wake widens. Near the free-end of the cylinder, illustrated in the bottom image, the wake structure disappears and two vortices with large vorticity magnitudes are seen.

When the cylinder has a small aspect ratio, the vortex shedding, which is suppressed near the free surface, dominates over nearly the entire length of the cylinder. However, there does remain a region of high vorticity at the free end, seen both in the vorticity fields and also demonstrated by the streamline visualizations that show the tip vortices coming off the bottom of the cylinders. For the cases of $AR = 2$ and 3, the aspect ratio is so small that a middle region does not exist, such that high vorticity from the free end dominates the flow behavior on the entire length of the cylinder. For each of these cases there exists a region of relatively high vorticity at the free end, from which tip vortices are shed. The marked drop in drag coefficient for the $AR = 2$ case is also a likely result of the vortex shedding coming entirely from the free end, combined with the flow propagating past the cylinder from below and around the two

sides.

5.4. Sectional Forces

Here, CFD predictions for the sectional forces along the length of a cylinder with $AR = 12$ are presented. A numerical mesh was generated with multiple patches (where a patch is a collection of boundary faces) along the length of the cylinder in order to investigate the loading on the cylinder as a function of its length and distance from the free surface. Each patch was a 3d cylindrical section. The patches were stacked vertically such that the eight patches comprise the entire geometry of the cylinder with $AR = 12$. It is possible to achieve spatial resolution as small as each face on the cylinder's surface. However, dividing the surface into eight patches achieved sufficiently detailed resolution in space, while also averaging out statistical noise.

The patch names are defined in Figure 16, where the portion of the cylinder that touches only air is named “air”. The patch named “Fountain” is located just above the still water line (SWL) but does touch water due to wave run-up on the front side of the cylinder. The patches that are located beneath the SWL are named “Water1” to “Water7” with increasing depth. The patches vary in length, such that patches near the ends are shorter, and patches along the middle span are longer. The exact length of each patch is provided in Figure 16.

Figure 16: Surface patch definitions for the cylinder with aspect ratio 12. The SWL sits at the base of the “fountain” patch. The patches under the SWL are labeled “Water 1” to “Water 7” with increasing depths.

Sectional drag coefficients were computed for each patch, where the projected frontal area depended on the corresponding patch location and force.

The time averaged sectional coefficients along the cylinder length are shown in Figure 17a. The drag coefficient is nearly uniform along the length of the cylinder. However, near the free surface the drag coefficient decreases rapidly, a trend that is consistent with previous studies found in the literature, e.g. Chaplin and Teigen (2003), Kawamura et al. (2002), Yu et al. (2008), Suh et al. (2011). Additionally, an

Figure 17: a) Sectional drag coefficient along the length of the cylinder. b) The normalized standard deviation per unit length of the fluctuating lift forces along the length of the cylinder. A peak in the standard deviation of the transverse force is seen near the middle of the cylinder span, with another smaller peak located near the free end.

increase in the drag coefficient can be seen at the bottom of the cylinder where the flow is altered due to the tip vortices from the free end.

Figure 18: Sectional drag coefficients from the present work are shown in red stars. Sectional drag coefficients from surface-piercing cylinders without free ends are shown in black, green, magenta, cyan and blue (Chaplin and Teigen (2003); Yu et al. (2008)). The dotted line indicates the location of the SWL.

The present work is compared against drag coefficient results for surface-piercing cylinders in Figure 18. Numerical results from the present work are shown in red stars. Data from Yu et al. (2008) and Chaplin and Teigen (2003) in the same vicinity of Fr , but larger Re , as the present work are also plotted. The sectional drag coefficients from the present work show the same trend from the top of the cylinder to nearly the bottom. At the bottom the drag coefficient increases, unlike the values from the previous work where the bottom of the cylinder was attached to a floor or a splitter plate, instead of being a free end. It is also shown that the predicted C_d values vary with dependence on both Re and Fr .

The standard deviation per unit length of the fluctuating lift force was calculated for each patch along the length of the cylinder, normalized by the maximum value. The results are presented in Figure 17b. A peak in the magnitude of the oscillatory lift force signal is seen near the middle of the cylinder's length. The magnitude of the fluctuating lift force is smaller near the free surface and near the free end, though not at the free end. At the end of the cylinder a local maximum in the lift force magnitude is seen, likely due to tip vortices shedding off of the free end. The relative magnitude of the lift force oscillation is indicative of the degree of vortex shedding. The standard deviations given on the right in Figure 17 match well with the visualizations of the vorticity present in Figure 14. The computed standard deviations of the sectional fluctuating lift force reiterate the presence of vortex shedding in the middle span of the cylinder, the lack of vortices near the free surface and lastly, the tip vortices from the free end, which were all seen in the visualizations of vorticity in Figure 14.

Next, FFTs of the sectional transverse forces were computed, and are presented as a function of the St in Figure 19. The largest peaks are seen for patches "Water2", "Water3" and "Water4", the three patches located along the middle span of the cylinder. This is the region where the vorticity field illustrated the greatest degree of vortex shedding, as well as the region where the peak in the standard deviation of the transverse force was found. Furthermore, these peaks are found at $St = 0.2$, the expected value for cylinders at the Re studied in this work.

Reduced peaks around $St = 0.2$ can be seen for the patches "Water1" and "Water5", the patches that surround the middle span of the cylinder. These are the regions that mark the beginning and the end of the region of strong vortex shedding. The FFTs as well as the standard deviation of the transverse force indicate that these regions see periodic vortex shedding, but that the vortices are smaller in magnitude than those between "Water1" and "Water5". Finally, the patches near the free-end of the cylinder, "Water6" and "Water7", show small peaks just below $St = 0.2$, as well as larger peaks closer to $St = 0.05$. The "Water7" patch gives the larger peak at the low St , which alongside the streamline and vorticity visualizations, suggests the tip vortices shed periodically and at a lower frequency than the vortices in the main wake.

The St number as a function of cylinder height is shown in Figure 20, against data presented by Farivar (1981); Fox and West (1993). In Farivar's work the free end was located at the top, so the data have been mirrored for comparison to the present work which modeled the free end at the bottom of the cylinder. The normalized height of zero represents the location of the free surface in the present work. Again, the work by Farivar (1981) did not include a free surface. The free ends are located at a normalized height equal to negative one. Fox and West's work studied a cantilevered cylinder attached to a wall Fox and West (1993). The free end of the cylinder in their work corresponds to the normalized height of negative one in Figure 20. The general trends in St as a function of distance from the free end

Figure 19: FFTs of the transverse force signal from each of the patches, from the fountain down to the water 7 patch at the bottom. Larger peaks in the FFT of the transverse force are observed for patches around the middle span of the cylinder (water 2, 3 and 4), indicating where the periodic vortices are largest and most pronounced.

Figure 20: The Strouhal as a function of cylinder height, compared against results from Farivar (1981); Fox and West (1993) Good agreement is found in the trends of St as a function of cylinder length.

of the cylinder match well. St numbers near 0.2 are observed for the end furthest from the free end, while near the free end, the St numbers are much lower. The flow travelling over the free end interferes with the Karman vortex shedding, and leads to a reduction in the shedding frequency in that region.

6. Summary and Conclusions

CFD simulations of flow past surface-piercing cylinders with aspect ratios from about 1 to 19 were performed at $Re = 2900$ and $Fr = 0.65$. The drag forces predicted from CFD simulations were compared against experimental measurements, across a wide range of aspect ratios from 1 to 19, and good agreement

was found. It was found that the combined effects of a free surface and a free end in the water affected the magnitude and frequency of loads on the cylinder, with a dependency on the aspect ratio.

At higher aspect ratios the drag coefficient was found to be nearly constant with a value near 0.75 from numerical simulations, and 0.85 from experimental work. A sharp drop in the drag coefficient was observed at $AR = 2$, as the flow can travel under as well as around the cylinder. Tip vortices were seen to dominate at small aspect ratios, while more pronounced vortex sheets were observed behind cylinders with larger aspect ratios. The Strouhal number decreased with the aspect ratio such that for the cases of $AR = 3$ and below, the lift force exhibited almost no periodicity at all, as illustrated by the lack of peaks in the FFTs of the transverse forces.

Sectional load investigations, a new contribution to studies of the combined effects of free surface and free ends, further corroborated flow field visualizations. The magnitude of the lift force was shown to peak near the middle of the cylinder span, with a local maximum at the free end where tip vortices shed. Near the free surface the magnitude of the lift forces was shown to die off entirely, as a result of the suppression of vortices near the free surface. Reduced Strouhal numbers were found for locations near the free end, indicating a different frequency of vortex shedding from the tip than from the center span of the cylinder.

The presence of a free-surface and a truncated cylinder end affects the structural loading, in both magnitude and frequency. These alterations to the loading behavior are dependent on the aspect ratio, such that at small depths of penetration, the drag is greatly diminished and the periodicity of loads from vortex shedding is absent. Understanding the flow behavior around bodies in the presence of these factors is of great importance to many engineering applications, such as offshore structures. The design of offshore structures should consider the implications of these factors of drag and frequency of loads.

Acknowledgment

The authors would like to acknowledge the support of a grant from the Department of Energy, award number DE-AC36-08GO28308 as well as support from the Massachusetts Clean Energy Center. This work is partially supported by the NSF-sponsored IGERT: Offshore Wind Energy Engineering, Environmental Science, and Policy (Grant Number 1068864). This work used the Extreme Science and Engineering Discovery Environment (XSEDE), which is supported by National Science Foundation grant number OCI-1053575. The authors acknowledge the Texas Advanced Computing Center (TACC) at The University of Texas at Austin for providing high performance computing resources that have contributed to the research results reported within this paper. <http://www.tacc.utexas.edu> for more information.

References

- Achenbach, E., 1968. Distribution of local pressure and skin friction around a circular cylinder in cross-flow up to Re 5e6. *J. Fluid Mechanics* 34, 625–639.
- Achenbach, E., Heinecke, E., 1981. On vortex shedding from smooth and rough cylinders in the range of reynolds numbers 6×10^3 to 5×10^6 . *Journal of Fluid Mechanics* 109, 239–251.
- Adaramola, M., Akinlade, O., Sumner, D., Bergstrom, D., Schenstead, A., 2006. Turbulent wake of a finite circular cylinder of small aspect ratio. *Journal of Fluids and Structures* 22, 919–928.
- Afgan, I., Moulinec, C., Laurence, D., 2006. Large eddy simulation of flow over a vertically mounted finite cylinder on a flat plate. In: *The 13th International Conference on Fluid Flow Technologies, Conference on Modelling Fluid Flow (CMFF06)*. Budapest, Hungary.
- Akilli, H., Rockwell, D., 2002. Vortex formation from a cylinder in shallow water. *Physics of Fluids* 14, 2957–2967.
- Breuer, M., 2000. A challenging test case for large eddy simulation: high Reynolds number circular cylinder flow. *International Journal of Heat and Fluid Flow* 21, 648–654.
- Catalano, P., Wang, M., Iaccarin, G., Moin, P., 2003. Numerical simulation of the flow around a circular cylinder at high Reynolds numbers. *International Journal of Heat and Flow* 24, 463–469.
- Chaplin, J., Teigen, P., 2003. Steady flow past a vertical surface-piercing circular cylinder. *Journal of Fluids and Structures* 18, 271 – 285.
- DeBar, R., 1974. Fundamentals of the KRAKEN code. Technical Report, Lawrence Livermore National Lab UCIR-760.
- Delany, K., Sorenson, E., 1953. Low-speed drag of cylinders of various shapes. *Nat. Adv. Comm. Aero.* 3038.
- E. Berberovic, E., van Hinsberg, N., Jakirlicacute, S., Roisman, I., Tropea, C., 2009. Drop impact onto a liquid layer of finite thickness: Dynamics of the cavity evolution. *Phys. Review. E* 79.
- Farivar, D., 1981. Turbulent Uniform Flow around Cylinders of Finite Length. *AIAA Journal* 13 (3), 275–281.
- Fox, T., West, G., 1993. Fluid-induced loading of cantilevered circular cylinders in a low-turbulence uniform flow. Part 1: Mean loading with aspect ratios in the range of 4 to 30. *Journal of Fluids and Structures* 7, 1–14.
- Frohlich, J., Rodi, W., 2004. LES of the flow around a circular cylinder of finite height. *J. Heat and Fluid Flow* 25, 537–548.
- Fung, Y., 1960. Fluctuating Lift and Drag Acting on a Cylinder in a Flow at Supercritical Reynolds Numbers. *Journal of the Aerospace Sciences* 27.
- Goncalves, R., Franzini, G., Rosetti, G., Meneghini, J., Fajarra, A., 2014. Flow around circular cylinders with very low aspect ratio. *Journal of Fluids and Structures*.
- Grigoriadis, D., Sarris, I., Kassinos, S., 2010. MHD flow past a circular cylinder using the immersed boundary method. *Computers and Fluids* 39, 345–358.
- Hay, A., 1947. Flow about semi-submerged cylinders of finite length.
- Inoue, M., Baba, N., Himeno, Y., 1993. Experimental and Numerical Study of Viscous Flow Field Around an Advancing Vertical Circular Cylinder Piercing Free-Surface. *J Kansai Soc Naval Arch* 220, 57–65.
- Iungo, G., Pii, L., Buresti, G., 2003. Experimental investigation on the aerodynamic loads and wake flow features of a low aspect-ratio circular cylinder. *Fluids Struct.* 28, 279–291.
- James, W., Paris, S., Malcolm, G., 1980. Study of viscous cross flow effects on circular cylinders at high Reynolds numbers. *AIAA Journal* 18, 1066–72.
- Kallinderis, Y., Ahn, H., 2005. Incompressible NavierStokes method with general hybrid meshes. *Journal of Computational Physics* 210, 75–108.
- Kawamura, T., Hiwada, M., Hibino, T., Mabuchi, I., Kumada, M., 1984. Flow around a Finite Circular Cylinder on a Flat Plate. *Bulletin of JSME* 27, 2142–2151.

- Kawamura, T., Mayer, S., Garapon, A., Srensen, L., 2002. Large eddy simulation of a flow past free surface piercing circular cylinder. *ASME Journal of Fluids Engineering* 124, 91–101.
- Kovasznyai, L., 1949. Hot-Wire Investigation of the Wake Behind Cylinders at Low Reynolds Numbers. *Proc. Roy. Soc.* 198, 174.
- Krajnovic, S., 2011. Flow around a tall finite cylinder explored by large eddy simulation. *J. Fluid Mech.* 676, 294–317.
- Lee, T., Lin, C., Friehe, C., 2007. Large-eddy simulation of air flow around a wall-mounted circular cylinder and a tripod tower. *J. Turbul.* 29, 1–28.
- Liang, C., Papadakis, G., 2007. Large eddy simulation of pulsating flow over a circular cylinder at subcritical Reynolds number. *Computers and Fluids* 36, 299–312.
- Lindsey, M., 1923. Drag of Cylinders of Simple Shapes. Report No. 619.
- Luo, S., Gan, T., Chew, Y., 1996. Flow around a Finite Circular Cylinder on a Flat Plate. *J. Wind. Eng. Ind. Aerodyn* 59, 69–93.
- Lysenko, D., Ertesvåg, I., Rian, K., 2013. Modeling of turbulent separated flows using OpenFOAM. *Computers and Fluids* 80, 408–422.
- Majumdar, S., Rodi, W., 1989. Three-dimensional computation of flow past cylindrical structures and model cooling towers. *Build. Environ.* 24, 3–22.
- Nichols, B., Hirt, C., 1975. Methods for calculating multi-dimensional transient free-surface flows past bodies. Technical Report, Los Alamos National Lab.
- Niedzwecki, J., Duggal, A., 1992. Wave Runup and Forces on a Circular Cylinder in Regular and Random Waves. *Journal of Waterway, Port, Coastal and Ocean Engineering* 118, 615–620.
- Noh, P., Woodward, P., 1976. SLIC (Simple Line Interface Calculation). 5th International Conference on Fluid Dynamics 59.
- Okamoto, S., Sunabashiri, Y., 1992. Vortex shedding from a circular cylinder of finite length placed on a ground plane. *J. Fluid EngT ASME* 114, 512–521.
- Okamoto, S., Yagita, M., 1973. The experimental investigation on the flow past a circular cylinder of finite length placed normal to the plane surface in a uniform stream. *Bull. JSME* 16, 805–814.
- Palau-Salvador, G., Stoesser, T., Frohlich, J., Kappler, M., Rodi, W., 2010. Large eddy simulations and experiments of flow around finite-height cylinders. *Flow Turbul. Combust.* 84, 239–275.
- Park, C., Lee, S., 2002. Flow structure around a finite circular cylinder embedded in various atmospheric boundary layers. *Fluid Dynamics Research* 30, 197–215.
- Pattenden, R., Bressloff, N., Turnock, S., Zhang, X., 2006. Unsteady simulations of the flow around a short surface-mounted cylinder. *Intl J. Numer. Meth. Fluids* 53, 895–914.
- Pattenden, R., Turnock, S., Zhang, X., 2005. Measurements of the flow over a low-aspect-ratio cylinder mounted on a ground plane. *Experiments in Fluids* 39, 10–21.
- Perkovic, L., Silva, P., Ban, M., Kranjcevic, N., 2013. Harvesting high altitude wind energy for power production: The concept based on Magnus effect. *Applied Energy* 101, 151–160.
- Relf, E., Simmons, F., 1924. The frequency of eddies generated by the motion of circular cylinders through a fluid. *Aero. Res. Council* 917.
- Roh, S., Park, S., 2003. Vortical flow over the free end surface of a finite circular cylinder mounted on a flat plate. *Experiments in Fluids* 34, 63–67.
- Rosetti, G., Vaz, G., Hoekstra, M., Goncalves, R., Fajarra, A., 2013. CFD calculations for free-surface-piercing low aspect ratio circular cylinder with solution verification and comparison with experiments. In: 32nd International Conference on Ocean, Offshore and Arctic Engineering. Nantes, France.

- Roshko, A., 1961. Experiments of the flow past a circular cylinder at very high Reynolds number. *Journal of Fluid Mechanics* 10, 345–356.
- Rostamy, N., Sumner, D., Bergstrom, D., Bugg, J., 2012. Local flow field of a surface-mounted finite circular cylinder. *Journal of Fluids and Structures* 34, 105–122.
- Rusche, H., 2002. Computational Fluid Dynamics of Dispersed Two-Phase Flows at High Phase Fractions. Dissertation, Imperial College of Science, Technology and Medicine.
- Sabersky, R., Acosta, A., Hauptmann, E., 1998. *Fluid Flow: A First Course in Fluid Mechanics*. MacMillan Publishers, New York.
- Sakamoto, H., Arie, M., 1983. Vortex shedding from a rectangular prism and a circular cylinder placed vertically in a turbulence boundary layer. *Journal of Fluid Mechanics* 126, 147–165.
- Sakamoto, H., Oiwake, S., 1984. Fluctuating forces on a rectangular prism and a circular cylinder placed vertically in a turbulent boundary layer. *J Fluid Eng-T ASME* 106, 160–166.
- Sarpkaya, T., 1977. Transverse Oscillations of a Cylinder in Uniform Flow. Rep. Monterey: Department of the Navy.
- Sarpkaya, T., Isaacson, M., 1981. *Mechanics of Wave Forces on Offshore Structures*. Van Nostrand Reinhold Company, New York.
- Schewe, G., 1983. On the force fluctuations acting on a circular cylinder in cross flow from subcritical up to transcritical Reynolds numbers. *J Fluid Mechanics* 133, 265–85.
- Singh, S., Mittal, S., 2005. Flow past a cylinder: shear layer instability and drag crisis. *Int J Numer Meth Fluids* 47, 75–98.
- Song, C., Yuan, M., 1990. Simulation of Vortex-Shedding Flow About a Circular Cylinder at High Reynolds Numbers. *Journal of Fluids Engineering* 112, 155–163.
- Sorenson, N., Bechmann, A., Zahle, F., 2011. 3D CFD computations of transitional flows using DES and a correlation based transition model. *Wind Energy* 14, 77–90.
- Spalart, P., Allmaras, S., 1992. A One-Equation Turbulence Model for Aerodynamic Flows. In: American Institute of Aeronautics and Astronautics (AIAA) Paper 92-0439. Reston, VA.
- Spalding, D., 1961. A Single Formula for the Law of the Wall. *Trans. ASME., J. Appl. Mech.* 28, 444 – 458.
- Suh, J., Yang, J., Stern, F., 2011. The effect of air-water interface on the vortex shedding from a vertical circular cylinder. *ASME Journal of Fluids Engineering* 27, 1 – 22.
- Sumner, D., 2013. Flow above the free end of a surface-mounted finite-height circular cylinder: A review. *Journal of Fluids and Structures*.
- Sumner, D., Heseltine, J., Dansereau, O., 2004. Wake structure of a finite circular cylinder of small aspect ratio. *Experiments in Fluids* 37, 720 – 730.
- Taniguchi, S., Sakamoto, H., Arie, M., 1981. Flow around circular cylinders of finite height placed vertically in turbulent boundary layers. *Bull. JSME* 24, 37–44.
- Travin, A., Shur, M., Strelets, M., Spalart, P., 1999. Detached Eddy Simulations Past a Circular Cylinder. *Flow, Turbulence and Combustion* 63, 293–313.
- Triton, D., 1959. Experiments on the flow past a circular cylinder at low Reynolds number. *Journal of Fluid Mechanics* 547.
- Weller, H., Tabor, G., Jasak, H., Fureby, C., 1998. A Tensorial Approach to Computational Continuum Mechanics Using Object-oriented Techniques. *Comput. Phys.* 12, 620 – 631.
- Wieselsberger, C., Betz, A., Prandtl, L., 1923. Versuche uber den Widerstand gerundeter und kantiger Korper. *Ergebnisse AVA Gottingen*, 22–32.
- Wu, C., Young, D., 2014. Simulations of free-surface flows with an embedded object by a coupling partitioned approach. *Computers and Fluids* 89, 66–77.

- Yu, G., Avital, E., Williams, J., 2008. Large eddy simulation of flow past free surface piercing circular cylinders. ASME Journal of Fluids Engineering 130.
- Zdrakovich, M., 1997. Oxford University Press.