Wake dynamics of a 3D curved cylinder in oblique flows

Soonhyun Lee a, Kwang-Jun Paik a,*, Narakorn Srinil b

a Department of Naval Architecture and Ocean Engineering, Inha University, Incheon, South Korea
b School of Engineering, Newcastle University, Newcastle Upon Tyne, United Kingdom

A R T I C L E   I N F O
Article history:
Received 22 January 2020
Received in revised form 5 July 2020
Accepted 14 July 2020

Keywords:
Vortex-Induced Vibration (VIV)
Wake
Curved cylinder
Wake topology
Large Eddy Simulation (LES) criterion
Computational Fluid Dynamics (CFD)
Flow regime
Marine riser

A B S T R A C T
Three-dimensional numerical simulations were performed to study the effects of flow direction and flow velocity on the flow regime behind a curved pipe represented by a curved circular cylinder. The cylinder is based on a previous study and consists of a quarter segment of a ring and a horizontal part at the end of the ring. The cylinder was rotated in the computational domain to examine five incident flow angles of 0°–180° with 45° intervals at Reynolds numbers of 100 and 500. The detailed wake topologies represented by $k_2$ criterion were captured using a Large Eddy Simulation (LES). The curved cylinder leads to different flow regimes along the span, which shows the three-dimensionality of the wake field. At a Reynolds number of 100, the shedding was suppressed after flow angle of 135°, and oblique flow was observed at 90°. At a Reynolds number of 500, vortex dislocation was detected at 90° and 135°. These observations are in good agreement with the three-dimensionality of the wake field that arose due to the curved shape.

 Peer review under responsibility of Society of Naval Architects of Korea. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

© 2020 Production and hosting by Elsevier B.V. on behalf of Society of Naval Architects of Korea.

1. Introduction

Limited oil and gas resources are being depleted onshore and in shallow offshore areas, so oil-producing regions are moving to deeper regions. This increases the use of floating offshore structures and requires risers to connect with subsea installations. Generally, offshore risers have a round cross-section and are long in the longitudinal direction, which results in non-linear motion characteristics. The representative phenomenon that leads the complex motion is Vortex-Induced Vibration (VIV) associated with the dynamic response of structures in fluids.

When ocean currents pass a riser, two free shear layers are developed at the stagnation point of the riser’s surface. Shear layers fall off asymmetrically and generate fluctuating lift forces in the crossflow direction and the drag forces in the flow direction (Blevins, 1990). These forces are small but persistent, which can result in fatigue problems, especially at the Touch-Down Point (TDP) or hang-off point. Furthermore, risers are not simple linear structures but have various configurations such as lazy-S or steep waves (Anderson and O’Connor, 2012). More complex flows around the riser can be expected, especially when a curved shape and multiple risers are used together.

It is possible to examine the flow regimes due to the curved cylinder by considering the basic concepts of wake dynamics that occur behind straight and inclined cylinders. Using a circular cylinder, an experimental study found a fluctuating velocity generated by separated shear layers (Bloor, 1964). This altering velocity causes a transition wave, which leads to an unstable wake. Wei and Smith (1986) studied these transition waves and observed secondary vortices through hydrogen-bubble flow visualization. The strength of the shear layer was quantified as an intermittency factor, which intermittency according with the Reynolds number (Prasad and Williamson, 1997).

For inclined cylinders, conventional two-dimensional assumptions have predominantly defined inherent flow characteristics, such as the shedding frequency, shed vortex filaments, and the drag force, which have simply been calculated with the Independence Principle (IP) (Ramberg, 1983; Schilching; Gersten, 2017). It is assumed that when a cylinder is inclined, normal components and not the tangential component can be used to derive the fluid forces on the structure. Ramberg (1983) experiment results showed that the IP can give a lower shedding frequency. Moreover, Lucor and Karniadakis (2003) mentioned that above a certain angle, the

---

* Corresponding author.
E-mail address: kwangjun.paiik@inha.ac.kr (K.-J. Paik).
Peer review under responsibility of Society of Naval Architects of Korea.

https://doi.org/10.1016/j.jinaoe.2020.07.005
2020 Production and hosting by Elsevier B.V. on behalf of Society of Naval Architects of Korea. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).
normal components are incorrect for an inclined cylinder at \( Re = 1000 \) compared with the value of the IP, which means the tangential component influences the wake dynamics.

The unstable flows around the circular cylinder can be defined as mode A and mode B, which are evidence that the flow around the cylinder is three-dimensional (Williamson, 1980, 1996). Through an autocorrelation of transverse velocity component \( v \), a large organized vortical structure was found to change depending on the inclined angle experimentally and numerically (Zhou et al., 2009; Zhao et al., 2009). This observation also emphasizes the three-dimensionality of the flow.

Razali et al. (2010) found a vortex dislocation on a 45° inclined cylinder and proposed that the secondary vortices along the span propagate by disturbing the large vortical structure. Recently, through a numerical analysis, the features of wake at high Reynolds numbers have been studied to look at three-dimensional properties such as the change of vortex formation length and vortex distortions at sub-critical Reynolds numbers (Lehmkuhl et al., 2014). Since it is possible to do simulations at \( Re > 5000 \), it has been clear that the limitations of the IP affect not only the slope of the cylinder but also the results at higher Reynolds numbers due to extended three-dimensional behavior (Alijure et al., 2017).

Experiments were done with low Reynolds number with a ring and curved-shaped bluff body (Bearman, 1988). Miliou et al. (2003) considered a quarter segment of a ring where the radius of curvature \( R \) was 12.5D, where \( D \) is the diameter of the cylinder. In their study, when the flow was normal to the plane of curvature, the wake was similar to that of a straight two-dimensional cylinder. However, in a convex configuration parallel to the plane of curvature, vortex shedding was suppressed above a certain angle due to the weak incoming flow vector in the sectional planes.

When it comes to IP, each section of the cylinder along the span should have a different frequency because the inflow velocity varies for each part. However, in this case, the shedding of the top of the cylinder (\( St = 0.1761 \)) is dominant. Research has shown that vortex cores are more energetic through a comparison of \( \lambda_2 \) criterion (Jeong and Hussein, 1995) in the convex case, and it is suggested that in-phase shedding becomes closer to cylinder as the Reynolds number increases. On the other hand, it was observed that the vortex shedding is completely suppressed in the case of the inner configuration. In terms of \( \lambda_2 \)-definition, it can be led by dismissing the viscous effects and unsteady straining from the symmetric part of the gradient of the Navier-Stokes equations. This definition identifies the vortex cores by capturing the presence of a local pressure minimum in a plane perpendicular to the vortex axis, and it represents the center filament of rotation as a form of the vortex tube.

In the case of \( Re = 500 \), a Strouhal frequency (\( St = 0.1123 \)) similar to the two-dimensional shedding was found (Miliou et al., 2007). With the same geometry and the free-stream direction parallel to the plane of curvature of the cylinder, a subcritical regime at \( Re = 3900 \) was studied, which showed that a recirculation region was suppressed at an inclination angle of 45°. As a result, the IP only has good agreement at the upper part of the cylinder and low Reynolds number, and the basic wake structure is divided into an upper wake of a regular vortex and a lower wake, which increases the three-dimensionality. Furthermore, in the same way as low Reynolds numbers, it was found that one single shedding frequency (\( St = 0.2197 \)) was dominant on the entire cylinder (Gallardo et al., 2013, 2014).

To avoid the geometrical effect on the curve cylinder, several studies have been done with modified geometries or different flow directions. At \( Re = 100 \), the effect of extending the top of the cylinder in the concave configuration was investigated. Vortex shedding was generated in the upper extended part in the same way as a straight cylinder, but there was no big difference in the curved part’s wake region (Miliou et al., 2003). Jiang et al. (2018) proposed that the vertical extension is Reynolds dependent by comparing free-stream velocities at several different Reynolds numbers (\( Re = 100 \) and 500). In the case of the concave direction (incident flow angles is 180°) at \( Re = 3900 \), it was suggested that an extension of 6D is needed to prevent the upper free-slip phenomenon, which suppresses the vertical velocity caused by the symmetry boundary condition. In this case, the straight and curved sections had distinct shedding frequencies of \( St = 0.208 \) and 0.223, respectively (Gallardo et al., 2013).

One of the factors affecting riser motion is the direction of flow. Previously, numerical calculations with a cylinder that was the same as that of Miliou et al. (2007) were studied in various directions by Jung et al. (2019). They focused on the comparison of the sectional fluid force coefficient by section and the vortex core around the riser according to the directions of five incident flow angles (0°, 45°, 90°, 135°, and 180°) at \( Re = 100 \). It was found that the vortex shedding was suppressed when the angle exceeded a certain number, and the sectional fluid forces showed a change due to the three-dimensionality of the wake. Therefore, it was suggested that the interaction between the axial flow and the wake greatly affects the incident flow angle.

In this study, the main objective is to investigate the change of flow characteristics due to the flow direction with two different Reynolds numbers. One is a laminar flow represented by \( Re = 100 \) and the other is a transient flow in \( Re = 500 \). To have more vigorous vortical structures, a large eddy simulation (LES) was applied as a turbulent model to describe smaller eddies as much as possible, and various aspects were tested to minimize the calculation instability. As the flow velocity increases, the flow is transferred to the turbulence region and is expected to display more complex vorticities. We tried to find out how the flow is altered when the velocity changes according to the flow direction with a curved cylinder. The \( \lambda_2 \)-criterion and vorticity contours were utilized to visualize the wake distribution. By interpreting these variables, the instability of the wake field was studied. It is obvious that the force and shedding frequency are influenced by the wake field. Therefore, to obtain the relationship of the acting force on the cylinder and wake topology, the integrated force coefficient and frequency were evaluated.

2. Numerical methodology

2.1. Governing equations

The governing equations for fluid flow are known as the Navier-Stokes (N–S) equations, which consist of the mass conservation equation and the momentum conservation equations. For incompressible N–S equations that describe incompressible viscous flow,

\[
\nabla \cdot \mathbf{U} = 0
\]

\[
\frac{\partial \mathbf{U}}{\partial t} + (\mathbf{U} \cdot \nabla) \mathbf{U} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{U}
\]

where \( \mathbf{U} \) is the fluid velocity, \( p \) is the pressure of the fluid, \( \rho \) is the fluid density, and \( \nu \) is the kinematic viscosity. As the N–S solver, we used ANSYS Fluent Version 18.1, which can simulate three-dimensional computations.

At \( Re = 100 \), laminar conditions were applied, and LES was applied to describe the transient flow, which is represented as \( Re = 500 \). LES resolves only the large eddies using a transient
calculation, while the small-scale eddies are modeled rather than resolving the whole spectrum of turbulent scales using a method known as Direct Numerical Simulation (DNS). The principle idea of LES is to reduce the cost of practical engineering problems by extracting the smallest length scales. However, LES still requires sufficiently finer meshes than a typical Reynolds Averaged Navier-Stokes (RANS) model, which leads to a long calculation time (CPU time) and uses much memory (RAM).

The governing equation for LES is obtained by filtering the time-dependent \( \text{N} \) \( \text{e} \) \( \text{s} \) equations. The eddies that are smaller than the filter width or grid spacing are filtered out. A filtered variable is defined as:

\[
\Psi(x) = \int_D \varphi(x') G(x, x') \, dx'
\]  

(6)

where \( D \) is the fluid domain, and \( G \) is a filter function that determines the scale of the resolved eddies. In this study, as a subgrid-scale (SGS) stress filter called the Dynamic Smagorinsky-Lilly

![Fig. 1. (a) Side view and (b) Oblique view with flow directions of the curved cylinder.](image)

![Fig. 2. Boundary conditions of the computational domain.](image)

<table>
<thead>
<tr>
<th>Table 1</th>
<th>Detailed information for all calculations.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Purpose</td>
<td>Case</td>
</tr>
<tr>
<td>Time step Test</td>
<td>T1</td>
</tr>
<tr>
<td></td>
<td>T2</td>
</tr>
<tr>
<td></td>
<td>T3</td>
</tr>
<tr>
<td></td>
<td>T4</td>
</tr>
<tr>
<td></td>
<td>T5</td>
</tr>
<tr>
<td>Prism layer Test</td>
<td>P1</td>
</tr>
<tr>
<td></td>
<td>P2</td>
</tr>
<tr>
<td></td>
<td>P3</td>
</tr>
<tr>
<td>Base size Test</td>
<td>Coarse</td>
</tr>
<tr>
<td></td>
<td>Medium</td>
</tr>
<tr>
<td></td>
<td>Fine</td>
</tr>
<tr>
<td>LES mesh dependency Test</td>
<td>RKE</td>
</tr>
<tr>
<td></td>
<td>LES-Coarse</td>
</tr>
<tr>
<td></td>
<td>LES-Medium</td>
</tr>
<tr>
<td></td>
<td>LES-Fine</td>
</tr>
<tr>
<td></td>
<td>LES-Long</td>
</tr>
<tr>
<td>Flow direction</td>
<td>Convex</td>
</tr>
<tr>
<td></td>
<td>Oblique (45°)</td>
</tr>
<tr>
<td></td>
<td>Perpendicular</td>
</tr>
<tr>
<td></td>
<td>Oblique (135°)</td>
</tr>
<tr>
<td></td>
<td>Concave</td>
</tr>
<tr>
<td></td>
<td>Convex</td>
</tr>
<tr>
<td></td>
<td>Oblique (45°)</td>
</tr>
<tr>
<td></td>
<td>Perpendicular</td>
</tr>
<tr>
<td></td>
<td>Oblique (135°)</td>
</tr>
<tr>
<td></td>
<td>Concave</td>
</tr>
</tbody>
</table>
model was chosen. The detailed explanation can be found in the ANSYS Fluent user guide.

In this study, a pressure-based segregated algorithm was applied to solve a pressure or pressure correction equation, which was derived by continuity and momentum equations. The pressure-based solver uses a sequential solution algorithm, which requires an iterative solution loop for the numerical solution to converge. In the segregated algorithm, the solution variables are solved by individual governing equations. This algorithm stores the discretized equations in memory one at a time and updates fluid properties based on the solution. The discretized governing equations are obtained by considering the unsteady conservation equation.

To obtain the cell value, the current study uses least squares cells for the gradient and a second-order method for pressure. The first-order upwind scheme, which sets the face value equal to the cell-center value of $\phi$, was applied for the transient formulation. For the momentum discretization, the second-order upwind scheme and bounded central differencing were used for laminar flow and LES.

To solve the N–S equations, we used the Semi-Implicit Method (SIMPLE) for LES calculations. This method applies a relationship between the velocity and pressure to obtain the pressure field and to carry out mass conservation. Additionally, SIMPLEC (SIMPLE-Consistent) was applied to accelerate the convergence in the calculation for laminar flow.

### 2.2. Computational domain

To validate the case and observe the differences due to the flow

<table>
<thead>
<tr>
<th>Case</th>
<th>$C_x$</th>
<th>Error rate (%)</th>
<th>$C_z$</th>
<th>Error rate (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Jung et al. (2019)</td>
<td>1.165</td>
<td>–</td>
<td>0.310</td>
<td>–</td>
</tr>
<tr>
<td>T1</td>
<td>1.099</td>
<td>5.67</td>
<td>0.296</td>
<td>4.52</td>
</tr>
<tr>
<td>T2</td>
<td>1.112</td>
<td>4.55</td>
<td>0.288</td>
<td>7.10</td>
</tr>
<tr>
<td>T3</td>
<td>1.125</td>
<td>3.43</td>
<td>0.293</td>
<td>5.48</td>
</tr>
<tr>
<td>T4</td>
<td>1.129</td>
<td>3.09</td>
<td>0.296</td>
<td>4.52</td>
</tr>
<tr>
<td>T5</td>
<td>1.130</td>
<td>3.00</td>
<td>0.297</td>
<td>4.19</td>
</tr>
<tr>
<td>P1</td>
<td>1.128</td>
<td>3.18</td>
<td>0.297</td>
<td>4.19</td>
</tr>
<tr>
<td>P2</td>
<td>1.133</td>
<td>2.75</td>
<td>0.302</td>
<td>2.58</td>
</tr>
<tr>
<td>P3</td>
<td>1.129</td>
<td>3.09</td>
<td>0.300</td>
<td>3.23</td>
</tr>
<tr>
<td>Coarse</td>
<td>1.136</td>
<td>2.49</td>
<td>0.298</td>
<td>3.87</td>
</tr>
<tr>
<td>Medium</td>
<td>1.133</td>
<td>2.75</td>
<td>0.298</td>
<td>3.87</td>
</tr>
<tr>
<td>Fine</td>
<td>1.138</td>
<td>2.32</td>
<td>0.301</td>
<td>2.90</td>
</tr>
</tbody>
</table>

Fig. 3. Mesh distribution of (a), (d) Convex case at Re = 100, (b), (e) at Re = 500, (c), (f) Oblique (135°) case at Re = 500.

Fig. 4. Mesh distribution of first cell height: (a) 0.001D, (b) 0.0001D.
Fig. 5. Comparison of wake topology depending on the mesh density and distribution at \( Re = 500, \lambda_2 = -0.8 \): (a)–(d) Side view, (e)–(h) Oblique view.
direction, we chose the same geometry as Miliou et al. (2007). The curved cylinder has a circular cross-section and is a quarter segment of a ring (torus). A part of the ring shape has a non-dimensional radius of curvature $R = D$ of 12.5, where $D$ is the diameter of the circular cross section, and $R$ is the radius between the center of the cylinder cross sections and the center of the ring. At the location of the end of the quarter segment, a horizontal extension, $L_h = 10D$, is applied to perform a calculation without end effects on the wake.

Fig. 1 (b) shows the geometries corresponding to the flow direction based on the center of the cylinder at $z = D = 0$, which is the main Cartesian coordinate. For the sectional approach, the non-dimensional arc length $s/D$ was considered, which is 19.6 at the end of the curved part. There are five directions of flow past the circular cylinder, which define with the angle of incident flow, $\Phi$. The flow is called a parallel flow when the flow direction is aligned with the plane of curvature of the cylinder. There are two parallel flows that are convex ($0^\circ$) when the flow is towards the outer face of the curved cylinder. A concave configuration ($180^\circ$) has flow past the inner face of the curved cylinder. When the uniform flow is perpendicular to the curvature of the cylinder, it is defined as a crossflow with $90^\circ$. Additionally, there are two oblique flows where the free-stream passes with angles of $45^\circ$ and $90^\circ$ (see Fig. 2).

The boundary conditions are defined below:

1) A velocity-inlet condition is assigned to the inflow plane with a uniform velocity profile without any perturbations; i.e., $u = U_\infty$, $v = 0$, and $w = 0$. 

Fig. 7. Comparison of wake topology at $Re = 500$, $z_2 = -0.8$: side view of (a) LES-fine, (c) LES-long, top view (b) LES-fine, (d) LES-long, mesh distribution: (e) LES-fine, (f) LES-long case.
2) At the outflow plane, a pressure-outlet condition known as the Neumann boundary condition is imposed for the velocity components \( \left( \frac{\partial u}{\partial n} = \frac{\partial v}{\partial n} = \frac{\partial w}{\partial n} \right) \), and the pressure is set to zero. This corresponds to a fully developed zero-stress condition.

3) The cylinder surface is treated as a stationary and non-slip wall; i.e., \( u = 0, v = 0 \), and \( w = 0 \).

4) A symmetry condition is applied in the remaining four planes.

Based on the rotating point of the cylinder, the computational domain consists of \(-46.5D \leq x \leq 53.5D, -30D \leq y \leq 30D, \) and \(-30D \leq z \leq 0D\). All calculations only rotate the cylinder around the rotation point in the same domain.

2.3. Mesh generation and distribution

The mesh for the computational calculations was generated by Star-CCM + version 13.06. In order to analyze fluid flows correctly, it is important to obtain an optimum mesh distribution through trial and error. Therefore, to conduct many case studies, we need to generate a mesh in a short time. Star-CCM + can quickly generate the grids and is advantageous for creating structured meshes that have a very regular grid structure. It is easy to create a prism layer that captures the boundary layer that develops when a fluid moves over a solid surface using a prism layer mesher. LES requires a fine mesh and a time-step that is sufficiently small to resolve the energy-containing eddies, which leads to a massive number of mesh elements and a long time for mesh generation.

The mesh resolution determines the fraction of turbulent kinetic energy that is directly resolved. Moreover, the wall-normal and wall-parallel spacing must be decreased to resolve smaller eddies. Ideally, it should have a dimensionless wall distance \( y^+ \) less than 1, which captures a realistic boundary layer through wall function. To achieve sufficient \( y^+ \), the near-wall thickness has been set as is 0.01D, and the number of prism layers has been set as 20. The exact mean values were 0.3 and 0.05 each for laminar and transient flow, respectively.

The grid settings are indicated in Table 1. As mentioned before, all the grid distributions are divided into four major parts that change according to the distance from the cylinder. Based on the base sizes \( 2.2D \) and \( 1.8D \) at \( Re = 100 \) and \( Re = 500 \), respectively, the fine part near the curved cylinder to resolve small eddies has a ratio of 10 percent of the base size, 25 percent of the medium size part, and 50 percent of the coarse part. Furthermore, a surface curvature of 36 points per circle is applied to create a smooth cylindrical surface (see Fig. 3).

2.4. Convergence test

Generally, an increase in the number of meshes or a smaller time-table:

<table>
<thead>
<tr>
<th>Case</th>
<th>Re</th>
<th>( C_x )</th>
<th>Error rate (%)</th>
<th>( C_z )</th>
<th>Error rate (%)</th>
<th>St</th>
<th>Error rate (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Convex (Jung et al., 2019)</td>
<td>100</td>
<td>1.165</td>
<td>-</td>
<td>0.310</td>
<td>-</td>
<td>0.1650</td>
<td></td>
</tr>
<tr>
<td>Convex</td>
<td>1.138</td>
<td>2.1</td>
<td>0.301</td>
<td>2.8</td>
<td>0.1608</td>
<td>2.5</td>
<td></td>
</tr>
<tr>
<td>Concave (Miliou et al., 2007)</td>
<td>1.075</td>
<td>2.3</td>
<td>0.376</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>Concave</td>
<td>1.099</td>
<td>2.3</td>
<td>0.420</td>
<td>9.6</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>Convex (Miliou et al., 2007)</td>
<td>500</td>
<td>0.920</td>
<td>-</td>
<td>0.380</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>Convex</td>
<td>0.872</td>
<td>5.2</td>
<td>0.343</td>
<td>9.7</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>Concave (Miliou et al., 2007)</td>
<td>0.783</td>
<td>5.3</td>
<td>0.341</td>
<td>9.8</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
<tr>
<td>Concave</td>
<td>0.742</td>
<td>5.3</td>
<td>0.347</td>
<td>9.8</td>
<td>-</td>
<td>-</td>
<td></td>
</tr>
</tbody>
</table>

Fig. 8. Oblique view of wake topology along the flow directions at \( Re = 100 \). (a)–(e) Present calculation with nondimensional velocity contour, \( \lambda_2 = -0.1 \); (f)–(j) Results of Jung et al. (2019), value of \( \lambda_2 \) is unknown.
step gives more precise results. LES simulations are very sensitive to the grid resolution and the numerical method. A large number of cells leads to longer computation time. In spite of the various efforts to find a standard method for evaluating numerical uncertainty, there is no specific method currently verified by CFD. To have an adequate running time and number of meshes, grid-convergence studies were performed with different thicknesses of the prism layer, base sizes of the mesh, and time steps. As a reference, the results were compared to the calculations of a relatively recent CFD study (Jung et al., 2019).

A transient CFD analysis highly depends on the time step. When the time step was 0.05 s, the error rate of the force coefficient was 5.67% in the x-direction and 4.52% in the z-direction. The error rate was reduced to 3.00% and 4.19% respectively, in the T5 case. Thus, for further study considering the calculation time and reliability, the time step was chosen as 0.005 s for $Re = 100$. For $Re = 500$, we considered the recommended time-step calculation provided by ANSYS Fluent.

As shown in Table 2, three different cell heights were evaluated, and a cell height of 0.001D showed the lowest error rate. It was observed that the smallest height of the first cell is not the best option, and 0.001D was chosen for other cases. As shown in Fig. 4 (a), the end cell of the prism layer is not well mated with the structured grid. This mesh distribution can lead to a distorted result. However, the number of the minimum prism layers recommended in the LES calculation is 20. Therefore, in this study, we continued under the following conditions.

As one of the options for meshing, a base size that sets other values, such as the target size, minimum size, and the surface size, adjusts the overall grid density. In this study, there are three cases depending on the base size. Remarkably, the coarse case was similar to the fine case, which has the lowest error rate, rather than the finer case converging to one value. In terms of verification of calculation, the smaller base size that has a finer mesh distribution was chosen for further work.

2.5. LES mesh dependency test

As mentioned before, LES requires a sufficiently fine mesh to capture the turbulent vortical structures. The use of the $\lambda_2$ criterion is one of the best ways to present the vortex cores. In this section, the wake topologies are compared by creating a $\lambda_2$ criterion iso-surface at a value of $-0.8$ with five cases, as shown in Fig. 5 and Fig. 7. First, the wake topology was compared to define how LES and realizable k-epsilon (RKE) models capture the vortex core differently.

Fig. 5 (a) and (c) clearly show that the two turbulence models depict vortex cores differently, despite having the same number of
meshes. The RKE case illustrates a vortex structure that bends along the cylinder, similar to $Re = 100$. However, the LES-medium case has many complex wake topologies, which include shedding structures to a further distance from the cylinder and massive vortical structures. Additionally, this complexity also appears in the integrated x-direction force coefficient shown in Fig. 6. Based on this, the result of LES oscillates very unstable, but the averaged value follows the average value of the RKE case.

Secondly, we verified the difference of the vortical structure according to the density of the grid. In Fig. 5 (b)–(h), it is apparent that the denser the mesh is, the smaller the vortex cores that it describes are. As shown in Fig. 5 (a), the mesh used for the LES-coarse, medium, and fine cases has a boundary at the end of the fine mesh part. Therefore, it can be seen that the vortex core became blunt in Fig. 5 (d). Based on the results, to capture precise vortex cores, it is necessary to use a fine enough mesh.

Finally, we tried to find a vortex core that was omitted by a coarse grid by extending a fine mesh behind the cylinder, as shown in Fig. 7 (f). As a result, the vortex core can be observed up to the end of the computational domain. At the end of the domain, the vortex shedding vanished, which indicates that the length of the domain is sufficient.

Fig. 6 shows that the force coefficients are unstable without a tendency, but the oscillation is based on the same mean value. These results indicate that the x-direction force on the cylinder depends on the turbulence model and the mesh around the cylinder rather than the mesh extension of the fine mesh part. In further studies at $Re = 500$, the fine mesh that extends to the end of the domain was used to capture the wake topology sufficiently.

2.6. Validation and verification

The time-averaged force coefficient and Strouhal number are compared with the results of Miliou et al. (2007) and Jung et al. (2019) in Table 3. It shows similar forces acting on the convex and concave cases at $Re = 100$ except in the z-direction force in the concave case. There is a difference of the horizontal extended part, which is absent in the study by Miliou et al. (2007), the z-direction...
force is expected to be bigger. At $Re = 500$, the average force values show a difference within about 10%, which is similar to the result at $Re = 100$. Thus, it seems reasonable to use it as a calculation reference. Since both results are from a numerical analysis, it is reasonable to refer to them as references rather than an exact comparison.

The wake topologies with non-dimensional velocity are compared in Fig. 8, including convex and concave cases, in order to consider the incoming flow incidence angle. Similar to a previous study (Jung et al., 2019), it can be seen that the vortical structure is distorted near the curved part of the cylinder in the convex case, and there is suppression of vortex shedding and the vortex pairs generated at the upper part in the concave case. In addition, there is no vortex core generation in the horizontal part, but it appears at the curved part. As the incident angle increases to $90^\circ$, vortex shedding that is suppressed at the bottom of the curved cylinder at $0^\circ$ is generated throughout the cylinder. These vortical structures show good agreement in the flow visualization.

At $Re = 500$, Fig. 9 shows an iso-surface that has a value $l_2/\sqrt{C_0} = 0.8$ for the $l_2$. It is compared with the numerical and experimental observation of Miliou et al. (2007). Unlike the previous numerical vortical structure, more vigorous vortex core distributions are observed in the present simulation, as shown in Fig. 9 (a) and (d). Further away from the cylinder, the vortex core becomes wider. This difference appears to be due to the grid and the program used in the calculation, and the LES calculation shows a large difference in the wake depending on the size of the grid. In terms of the concave configuration, as shown in Fig. 9 (f)–(h), the vortex shedding is absent along the span like at Reynolds number 100, and unsteadiness in the wake topology at the top part of the cylinder is depicted.

The time-averaged $0.3D$ upstream velocity in the x- and z-directions along the span are shown in Fig. 10. These figures show good agreement of the velocity distribution except at higher $s/D$ in the concave configuration. This suggests the horizontal part of the concave cylinder leads to lower velocity. Although there is a difference between the average integrated force coefficient and the vortical structures, the present calculation is verified by the fact that the difference is acceptable and has a similar tendency.

3. Results: wake characteristics

3.1. Convex configuration ($0^\circ$)

Fig. 11 (a) and (b) show a three-dimensional wake topology in a convex configuration with two different Reynolds numbers. Two different $l_2$ criterion values were used to compare only the structure of the wake. It was observed that the vortex core was suppressed by the curved shape at lower velocity, which leads the lower part of vortex cores to bend toward the pipe. However, it is generated along the span when the velocity is increased, and we observed a topology of a ring shape that indicates threedimensionality around the horizontal part. As discussed in the study by Miliou et al. (2007), this structure shows the difference velocity depending on the section that represents the non-dimensional arc length, $s/D$. It is reasonably evident that the non-dimensional velocity is different along the span.

Fig. 11 (c), (d), (e), and (f) illustrate the spanwise vorticity at $z/D$ from the tops of cylinders 3 and 11, respectively, which show various flow regimes according to the in-flow velocity and ovality.

![Fig. 11. Comparison of wake topology: (a) $Re = 100, l_2 = -0.03$, (b) $Re = 500, l_2 = -3$ and z-direction vorticity; (c) $Re = 100$ at $z/D = 3$, (d) $Re = 500$ at $z/D = 3$, (e) $Re = 100$ at $z/D = 11$, (f) $Re = 500$ at $z/D = 11$.](image-url)
of the cylinder cross-section. Based on the slowest flow, Fig. 11 (c) shows a long shear layer around the cylinder, which generates large regular vortex shedding. Relatively small vortex shedding occurs more frequently, which is represented by a high Strouhal number, as shown in Table 3. The shedding becomes unstable when it is more distant from the cylinder. It is regarded that the recovery of the velocity leads the vortex core to be unstable.

At the lower part of the curved cylinder ($z/D = 11$), stationary counter-rotating vortex cores appear as long and straight due to the fast in-flow velocity, as shown in Fig. 11 (e). When the velocity is high, the shear layer shows a wavy shape in Fig. 11 (f). Vortex shedding is also generated with a large width, which represents the ring shape of the wake topology.

### 3.2. Concave configuration (180°)

Unlike the convex case, straightened stationary vortices were depicted in the streamwise direction rather than the vortex shedding not being found at low Reynolds number. Furthermore, the distribution of the $z$-direction vorticity contour supports this result. In Fig. 12 (e), a long stationary vortex pair was observed at $z/D = 3$. The lower part of the cylinder ($z/D = 11$) shows stable counter-rotating vortex cores, which is interpreted as the oval shape of the cross-section leading the flow to pass smoothly without shedding.

When the flow velocity increased, unstable vorticities were observed at the top of the curved cylinder, but clear vortex shedding was not observed. Rather than a regular vortex street, a pair of vortices was observed in the far wake. It is related to the oscillation to the $y$-direction, which was absent at $Re = 100$. Through a Fast Fourier Transform (FFT) of the lift force coefficient, the Strouhal number was observed as $St = 0.1184$, which is smaller than the result of a straight cylinder, $St = 0.2060$ (Jiang et al., 2018). This suggests that the curved shape and horizontal part cause attenuation of the vortex shedding and decrease the shedding frequency.

Fig. 12 (f) displays the vortices generated in the recirculation region, which leads to secondary flow. As a result, an additional vortex structure can be seen in Fig. 12 (d) after the shear layers. This observation indicates the flow is more energetic at high velocity and an increase of three-dimensionality. In terms of the flow direction, the convex and concave configurations show a significant difference of the flow regime and oscillation frequency at both Reynolds numbers. Therefore, the different wake characteristics show that it is meaningful to study various directions at various velocities.

---

**Fig. 12.** Comparison of wake topology of concave configuration: top view: (a) $Re = 100, \lambda_2 = -0.001$ (b) $Re = 500, \lambda_2 = -0.1$; side view: (c) $Re = 100, \lambda_2 = -0.001$, (d) $Re = 500, \lambda_2 = -0.1$; and $z$-direction vorticity: (e) $Re = 100$ at $z/D = 3$, (f) $Re = 500$ at $z/D = 3$, (g) $Re = 100$ at $z/D = 11$, (h) $Re = 500$ at $z/D = 11$. 
3.3. Perpendicular flow (90°)

Fig. 13 (a) – (d) show the top and side views of the wake topology with an incident flow angle of 90°. The vortex core at low velocity is shown to be delayed at the bottom of the cylinder, which is called oblique vortex shedding. This oblique alignment of the vortical structure causes unstable vibration of the cylinder shown in Fig. 14. Fig. 13 (e) and (g) illustrate that the vortex becomes weaker downward. Furthermore, at \(z/D = 11\), two vortex cores are observed, which generated a different primary vortex, as shown in Fig. 14 (a). This represents the three-dimensionality and complexity of the wake field because of the curved part of the cylinder.

When the flow velocity rises, the wake topologies are more complex and energetic. Compared to \(Re = 100\), the main difference is that there is a vortex dislocation that appears at \(s/D = 16\) and approximately 45.8°, as shown in Fig. 14. The vortical structure bends toward the vortex dislocation line. In addition, vortex shedding is suppressed at the end of the cylinder, which indicates the end effect. When it comes to the vorticity distribution, the upper vorticities (\(z/D = 3\)) are similar to the convex case. At \(z/D = 11\), like at a Reynolds number of 100, the z-direction velocity curl shows that the vortex core, which is separated from the main vortex core, occurs in the horizontal part. However, referring to Fig. 14 (b), these vorticities present the same line of the vortex structure, which is generated from the horizontal part.

To define the effect of this different wake topology, Table 4...
which represents the transitional and stable primary vortex core, similar to the cylinder, and no vortex dislocation is found. With the reverse flow regime shown in Fig. 15(f), the regular vortex street is generated on the leeward side, and a slanted phenomenon, the vorticity contours at the horizontal part in the perpendicular case are straightened due to the unstable velocity increases, shedding of the vortical structure is developed along the span until the middle of the horizontal part. Furthermore, Fig. 17 illustrates there is vortex dislocation at the initiation of the curved part, which implies there are two different shedding generated.

3.4. Oblique flow (45°)

When the flow past the curved cylinder with an oblique direction and incidence angle of 45°, the wake topologies depict both characteristics of convex and perpendicular cases simultaneously. As shown in Fig. 15 (a), the delayed shed vortex filaments in the horizontal part in the perpendicular case are straightened due to the angle of 45°. Although the vortex cores are disturbed at the end of the cylinder, it shows stable vortex shedding. As reflected in this phenomenon, the vorticity contours at z/D = 3 and 11 indicate a regular vortex street is generated on the leeward side, and a slanted elliptical cross section produces a long vortex core with similar shape. At Re = 500, a vortex structure is generated at an angle similar to the cylinder, and no vortex dislocation is found. With the stable primary vortex core, fine-scale vorticities are developed, which represents the transitional flow regime shown in Fig. 15 (f) and (b).

As shown in Table 5, the x-direction force fluctuates with two different dominant frequencies, which are due to the vortex core that occurs in the x-direction that is generated differently at the top and at the horizontal part by the curved shape. The frequency of the oscillation is almost double, and frequencies in other directions are similar to the smaller frequency considered as dominant frequency. However, the higher velocity denotes completely different frequencies.

The x-direction force oscillates with higher frequency, which implies the primary vortex shedding is generated with high frequency at Re = 500. The other frequencies show a relationship between the force and frequency. When the flow is fast, the y-direction force is low, and the frequency is high. When the z-direction force is high, the frequency is low. This shows that while the tilted flow velocity increases, the force on the y-axis of the entire cylinder decreases and increases in the z-direction.

3.5. Oblique flow (135°)

The simple topology of the wake was viewed when the flow incident angle is 135° with a Reynolds number of 100, as shown in Fig. 16. In terms of the vorticity, only the stationary vortex core, which is generated around the cylinder, is generated at two different sections. In addition, the evaluation of integrated forces presents that there is only a constant force rather than oscillating in any direction. In Fig. 16 (d), as the velocity increases, shedding of the vortical structure is developed along the span until the middle of the horizontal part. Furthermore, Fig. 17 illustrates there is vortex dislocation at the initiation of the curved part, which implies there are two different shedding generated.

The wake topology along the curved part is clearly observed as shown in Fig. 16 (b) and (d). However, an irregular and unstable vortex core is provoked at the horizontal part. This intermittency is reflected in the frequency of the integrated force coefficient shown in Table 6. The force in the y-direction was relatively steady (f_y,135deg = 1.740), which is predominantly caused by the curved part. This frequency is considered as the main frequency for this cylinder, which is also found in the x- and z-directions. The other two different frequencies were due to the unstable flow in the horizontal part. Consequently, with an angle of incidence of 135°, the vortex shedding that is suppressed at low velocity but occurs as the speed increased. Jung et al. (2019) predicted an angle at which
which angle-dependent shedding begins is also an important factor to predict the wake behind a curved cylinder.

**Table 5**  
Comparison of the x-, y- and, z-direction integrated force coefficient and frequency of x-, y- and z-direction force coefficient of oblique flow at 45° with two different Reynolds numbers.

<table>
<thead>
<tr>
<th>Re</th>
<th>Flow direction</th>
<th>$C_x$ mean</th>
<th>$f_x$ 45deg</th>
<th>$C_y$ mean</th>
<th>$f_y$ 45deg</th>
<th>$C_z$ mean</th>
<th>$f_z$ 45deg</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>45°</td>
<td>1.046</td>
<td>0.1648</td>
<td>-0.315</td>
<td>0.1642</td>
<td>0.124</td>
<td>0.1641</td>
</tr>
<tr>
<td>500</td>
<td></td>
<td>0.905</td>
<td>0.3295</td>
<td>-0.192</td>
<td>0.2098</td>
<td>0.282</td>
<td>0.1460</td>
</tr>
</tbody>
</table>

**Fig. 15.** Comparison of wake topology of oblique flow direction of 45°: top view: (a) $Re = 100, \lambda_2 = -0.03$ (b) $Re = 500, \lambda_2 = -2$; side view: (c) $Re = 100, \lambda_2 = -0.03$, (d) $Re = 500, \lambda_2 = -2$; and z-direction vorticity: (e)$Re = 100$ at $z/D = 3$, (f)$Re = 500$ at $z/D = 3$, (g) $Re = 100$ at $z/D = 11$, (h)$Re = 500$ at $z/D = 11$.  

Shedding occurs between perpendicular and concave configurations. It was suggested that the Reynolds number of the flow at
4. Conclusions

In this study, numerical studies have been investigated for the wake dynamics behind a curved cylinder with 5 different directions and two different Reynolds numbers. The curved cylinder had the same geometry as a previous study by Miliou et al. (2007), which consists of a quarter segment of a ring that has a radius of curvature of 12.5D and length of 10D for the horizontal part. By rotating the cylinders of the computational domain, flow at different incidence angles was implemented. To inspect the laminar and transient regimes in the flow, Reynolds numbers of 100 and 500 were adopted. LES was selected as a turbulent model for the vortical structures. For stability and reliability of calculations, convergence and mesh density tests were performed. First, the dependency of the time step was evaluated. The results from the smaller time steps

Fig. 16. Comparison of wake topology of oblique flow direction of 135°: top view: (a) Re = 100, \( l_z = 0.035 \), (b) Re = 500, \( l_z = 2 \); side view: (c) Re = 100, \( l_z = 0.035 \), (d) Re = 500, \( l_z = 2 \); and z-direction vorticity: (e) Re = 100 at \( z/D = 3 \), (f) Re = 500 at \( z/D = 3 \), (g) Re = 100 at \( z/D = 11 \), (h) Re = 500 at \( z/D = 11 \).

Fig. 17. Oblique view of wake topology with oblique flow at 135° and Re = 500.
with converged values were used in subsequent studies. The second variable was the first cell height with three different heights of the prism layer. The results showed that low or high height did not give the amenable results, which suggests it is necessary to find the optimum height.

Thirdly, the base size, which determines the density of the grid, was changed. The finer grid system had good agreement compared to previous computational results. However, owing to the fine grid resulting in long computation times, compromises were needed in the accuracy and economics of the numerical computation. In LES calculations that simulate large eddies, the dependence on the grid becomes larger. Lastly, to check the LES results according to the distribution of mesh, 5 cases were analyzed, including four different mesh distributions and the RKE turbulent model. To obtain reliable wake topologies, it was concluded that a long and fine mesh was required.

To understand the wake characteristics behind the cylinder, the $l_2$ criterion and z-direction velocity curl were applied to the wake field. The observed wake regions showed different wake dynamics according to the cylinder section and Reynolds number. Overall, the vorticity contour at the top of the cylinder ($z/D = 3$) exhibited flow regimes similar to that of a straight cylinder. In addition, the main vortex shedding occurred in the curved part. At a Reynolds number of 100, the vortex shedding was absent when the angle was higher than $135^\circ$. However, at $Re = 500$, shedding occurred even at $135^\circ$, and vibration was detected as a result of the instability of the flow at the top of the cylinder even in the concave configuration.

Vortex pairs generated by this vibration were observed, and the Strouhal number was 0.1184, which is smaller than that of a straight cylinder. Furthermore, an additional wake topology was revealed in the far wake. Consequently, the vortex shedding that was initiated by the curved cylinder depends on the Reynolds number and inflow angle. At a Reynolds number of 500, when the flow passes with angles of $90^\circ$ and $135^\circ$, vortex dislocation was found as a result of different vibrations in the curved part and horizontal part of the cylinder. The horizontal part shedding took place more slowly than the main shedding and showed deflection of the vortex core. This was not clearly detected at $45^\circ$, and it is considered that there is a part where the two parts begin to vibrate differently between $45$ and $90^\circ$.

Lastly, the z-direction vorticity distribution at $z/D = 11$ showed the shear layer differences produced by different elliptical sections due to the cylinder slope at $0^\circ$ and $180^\circ$. In the vertical case, a wide recirculation region was formed on the lee side, which produced a secondary flow. This is similar at $45^\circ$ and $135^\circ$, and it represents the three-dimensionality of the wake field increasing.

In conclusion, the wake topologies of oblique flow exhibit a combination of parallel and perpendicular flow regimes. It was clear that vortex shedding depends on the Reynolds number and free-stream flow direction. The curved circular cylinder leads to different flow regimes along the span, which shows a more complex wake due to the shape. This complexity arises from the axial flow caused by the curved shape, the different cross-section depending on the position, and different in-flow velocity. Therefore, as future work, a complex geometry such as a Lazy-S catenary riser should be studied to understand more realistic aspects.

### Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

### Acknowledgement

This research was funded and conducted under the Competency Development Program for Industry Specialists of the Korean Ministry of Trade, Industry and Energy (MOTIE), operated by the Korean Institute for Advancement of Technology (KIAT) (No. N0001287, HRD program for Korea-UK Global Engineer Education Program for Offshore Plant). This research made use of the Rocket High Performance Computing service at Newcastle University.

### References


