A Two-Dimensional Computational Study of a Single Right Trapezoidal Cylinder Subjected to Unsteady Laminar Flow Regime at Low Reynolds Number

Sodsai Lamtharn¹ and Monsak Pimsarn²*
¹,²Mechanical Engineering Department, Faculty of Engineering, King Mongkut’s Institute of Technology Ladkrabang, Chalonerkung Rd. Ladkrabang, Bangkok, 10520, Thailand

* Corresponding Author: monsak.pi@kmitl.ac.th

Abstract
A right trapezoidal cross-section cylinder is used as guided bars for a barred tee in the long pipeline and piping system to protect the moving of pigs across the branched pipe. The cylinder must have the integrity to withstand to fluid flows-induced vibration which concerns the synchronization between vortex shedding frequency of fluid flows with a natural frequency of the cylinder. Therefore, this article aims to numerically investigate the vortex shedding frequency responses which are in terms of Strouhal number, flow force coefficients and in order to provide additional data for use in the absence of more applicable data for engineering design of the single right trapezoidal cylinder in structural integrity point of view. In this study, the simulation of the single right trapezoidal cylinder immersed in the uniform oncoming laminar flow regime with low Reynolds number; \( Re = 100 \), was performed. There are various sharpening angles in this study; defined as angle which is measured from virtual line parallel with height base side to slant side or \( 90^\circ - \text{acute angle} \) of the cylinder, in the range of \( 0^\circ, 15^\circ, 22.5^\circ, 40^\circ \) and \( 60^\circ \), accompanying with the cylinder side ratio \( (B/A) \) which is the ratio of the longest base-to-height base side, in the range of \( 1, 2, 3, 4, 5, 6 \) and \( 7 \). In the simulation, the Direct Numerical Simulation (DNS) with a finite volume method, an incompressible flow and constant fluid properties were employed. The computational results were validated against the published data. The effect of the sharpening angle and side ratio on the response of the single right trapezoidal cylinder were presented and discussed. The Strouhal number and flow force coefficients at fully ten cycle of saturated flow have been calculated. The simulated results show that increasing of sharpening angle, the Strouhal number slightly increases whilst the root mean square lift coefficients significantly increases. Additionally, as the side ratio increases, the Strouhal number and the root mean square lift coefficients decreases.

Keywords: single right trapezoidal cylinder, unsteady flow, incompressible flow, laminar vortex shedding
1. Introduction

A right trapezoidal cross-section cylinder is used as guided cylinder bars for a barred tee in the long pipeline and piping system to control propel of pigs in desired partway line and to protect the moving of pigs across the branched pipe for various internal pipe activities, such as batching fluid product, cleaning, separating fluids and inspecting these systems. The pigs are a spherical or circular cylindrical shaped device, which can be pushed through the main pipe from the pigs’ launcher station by pressurized fluid flows through the desired main pipeline, branched pipe and together with all piping component, pipe supports, valves and inline instruments to the pigs’ receiver station. The main pipe is a major portion of the long pipeline and piping system that composes of many branch connections between the pig traps station. When the system is filled with pressurized liquid to start-up operation, the fluid flows past the main pipe and flowing into a branched pipe together with past over the barred tee. To control propel of pigs in the desired way and protect the moving of pigs across the branched pipe, the barred tees are provided as shown on figure 1 (left) and (right), the guided cylinder bars attached cross inside the branched pipe with fillet-welded joints at the connection point. Normally, the barred tee is installed on all branch pipes, with a diameter larger than half or one-quarter of the main pipe diameter.

![Figure 1](image_url)

**Figure 1.** (Left) Schematic of flow passing the main pipe, branched pipe and barred tee, (right) Details cross-section of the cylinder bar

As the flow passes over a barred tee which composes of a square, a rectangular and a right trapezoidal cylinder bar cross-section, the cylinder bar must have the integrity to withstand to fluids flow-induced vibration which concerns the synchronization between vortex shedding frequency with the natural frequency of the cylinder bar, in the aspect of engineering design. The synchronization vibration of them will cause the cylinder bar subjected to strong vibration eventually leading to fatigue damage. Then, the bars will detach from the branched pipe wall and flow into the main pipe. This causes the pigs stuck, valves face scraper or crash and the bars flow into other inline pieces of equipment, such as a pump impeller, a flow control valve, etc. Additionally, the strong cylinder bar vibration can result in pipe wall local stress amplification, especially, at the fillet-welded region.

Most of the research on flow past over a cylindrical bodies in the past has been intensively performed for circular, square, rectangular, equilateral triangle and equilateral trapezoidal cylinders rather than a right trapezoidal cylinder. For a comprehensive review on circular cross-section, the details are referred to the articles by Williamson [1] and Zdravkovich [2]. Mittal et al. [3] who carried out simulation study for flow past circular cylinder to investigate the effect of the blockage ratio at $Re \leq 150$, the blockage ratio $\gamma = 1\%$ and 5% with a stabilized space-time finite element formulation. Sohankar et al. [4, 5, 6] performed 2-D simulations for a square and a rectangular ($B/A = 1-4$) cylinder subjected to unsteady flow at approaching angle $\alpha = 0^\circ-90^\circ$, $Re \leq 200$ to study the effect of modeling parameters including the outlet of boundary condition. They employed the time step $\Delta t = 0.025$ and mesh size next to cylinder $\zeta = 0.004$ for accurate simulation. The proper values of $X_d \geq 25$ and $X_u = 11.1$ were utilized, where $X_u$ and $X_d$ are defined as dimensionless distance measured from both the inlet and outlet.
boundaries to the most upstream corner and the most downstream corner of the cylinder, respectively. At $B/A = 1$, the root mean square (RMS) lift coefficients, $CL_{RMS}$ and Strouhal number, $St$, increase with the increase of $Re$ up to the angle of incidence of $45^\circ$ after their decrease. A convective Sommerfeld boundary condition (CBC) is better than a traditional Neumann boundary condition (NBC) for flow around a square cylinder at non-zero an angle of approaching and the CBC more effective in reducing the CPU time, reducing the upstream influence from the outlet and reducing the necessary downstream extent. Sohankar et al. [6] performed numerical study and recommended for $Re < 100$ use proper value $X_d > 15$ units and for $Re = 100-200$ use $X_d > 10$ units. For CBC and the upstream influence of the outlet is effectively damped out on NBC at approximated $X_d > 25$ units. At $Re = 100 - 200$ with $\alpha = 0^\circ$, the blockage ratio $\gamma$ reduced from $5\%$ to $2.5\%$ and slightly decrease on the RMS lift coefficients and Strouhal number. Zhao et al. [7] provided the RMS lift coefficients and Strouhal number from the studied/measured influence of an angle of incidence for flow over a square cylinder subjected to VIV at $Re = 100$ with $\alpha = 0^\circ$, $45^\circ$, and $22.5^\circ$ and $\gamma = 2.5\%$. More additional studied for numerical simulations of 2-D unsteady flow past over a square cylinder at low and moderate Reynolds numbers are Sharma and Eswaran [8], Singh et al. [9], Sahu et al. [10] and Sen et al. [11]. Sohankar et al. [12], Robichaux et al. [13], Darekar and Sherwin [14] and Saha et al. [15] have been carried out for the simulation study of 3-D flow around a square cylinder at low and moderate Reynolds numbers, the reader is referred to the review these works. Okajima [16] conducted experimental on flow past over various rectangular cylinder in both a water tank and a wind tunnel to determine Strouhal number as a function of Reynolds number and the width-to-height ratio. The results show that the Strouhal number has a minimum value at the width-to-height ratio of 1:2 and a maximum value at 1:3. Okajima [17] and Okajima et al. [18] carried out simulation for flow past cylindrical rectangular cross-section and investigated the effect of blockage ratio on both oscillating and stationary rectangular cylinder. Arnab and Dalal [19] carried out 2-D numerical simulations of a triangular cylinder which placed in free-stream at $10 \leq Re \leq 250$ subjected to laminar flow with finite volume and more study on the RMS lift coefficients and Strouhal number. Alawadhi [20] numerically studied the 2-D incompressible laminar flow past over a triangular cylinder in a horizontal channel for a stationary and undergoing vertical oscillating motion with a fixed $Re = 100$. Jiang and Cheng [21] present a DNS study of the 2-D, 3-D on a square cylinder subjected to flow around in order to investigate the characteristics of wake flow and the force due to hydrodynamic at $Re = 10-400$. They found that a local maximum has been occurred at $Re = 168$, for the 2-D $St-Re$ relationship. Chung and Kang [22] studied effect height ratio of trapezoidal with laminar vortex shedding for $Re = 100$, 150 and 200. The result has shown that the Strouhal number for trapezoidal cylinders depends on the height ratio and $Re$.

The results indicate that when the cylinder’s natural frequency is equal to the Strouhal number of the stationary cylinder, the RMS lift coefficients reaches its maximum and at a high oscillating frequency that does not affect to the oscillating amplitude. As mentioned above from literature and in the past, the most applicable engineering data for flow past circular, elliptic, square, rectangular, triangle and trapezoidal cylinder are emphasis intense investigated but more absence data for the right trapezoidal cylindrical cross-section to use in barred tee engineering design. Therefore, this article aims to numerically investigate the vortex shedding frequency responses which are in term of Strouhal number, flow force coefficients and in order to provide additional these data for use in the absence of more applicable data for engineering design of the single right trapezoidal cylinder in structural integrity point of view.

2. Numerical model

The numerical details, numerical methods, governing equations, boundary conditions and grid structure for the flow past over the single right trapezoidal cylinder are discussed below.

2.1 Numerical details and grid structure

The geometry for this computational study of flow is shown in figure 2 (left), a stationary 2-D single right trapezoidal cylinder with projected width $d$ of the cylinder in the streamwise direction subjected to
unsteady laminar flow inlet with constant free-stream velocity $U_\infty$ flowing into side $A$ and $B$ at some flow approaching angle, $\alpha$. Here $B$ is the longest base side, $A$ is height base side, where sharpening angles $\beta$ is defined as angle which is measured from a virtual line parallel with height base side to slant side or (90° - acute angle°) of the cylinder. As depicted in figure 2 (left) the center of cylinder and force coordinates are placed on origin, the $L$ axis is the transverse direction and the lift force is positive, the $D$ axis is a streamwise direction and the drag force is positive. The projected width $d$ is used scale for length dimensional to dimensionless, to all force coefficients and Strouhal number, $St = fSd/U_\infty$, by $fS$ is the frequency of vortex shedding. Scaling with $U_\infty$ for all velocities and physical times are scaled with $d/U_\infty$. The blockage ratio which is confined boundary parameter is defined as $\gamma = d/H$, where $H$ is a vertical measurement from the upper wall to lower wall boundary of the computational domain.

In this study given $\beta$ varied in the range of $0^\circ$, $15^\circ$, $22.5^\circ$, $40^\circ$ and $60^\circ$ accompany with the cylinder side ratio, $B/A$ varied in the range of 1, 2, 3, 4, 5, 6 and 7. Some remaining key parameters came from the resultant studied work of Lamtharn and Pimsarn [23]. As re-written following, $Re = 100$, $d = 1$, $\alpha = 0^\circ$, $\gamma = 0.05$, $H = 20$, $X_0 = 10$ and $X_d = 15$, $\zeta = 0.004$. The non-uniform mesh distribution was used vary mesh size of 0.004-0.5 in an area that extends of 5 units all side from the nearest corner of the cylinder. The constant mesh size $\Delta = 0.5$, uniform distribution in the outside that area is employed, on each side of the cylinder is assigned 20 elements per unit length as mesh shown in figure 2 (right).

### 2.2 Boundary conditions

The inlet and outlet boundary are located at $X_0 = 10$ units upstream and $X_d = 15$ units downstream from the most upstream and the most downstream corner of the cylinder, respectively. The upper and lower boundary is located at $H/2$ of cylinder walls. In table 1, the boundary conditions for this computational study is shown, where $U = U_\infty$ and $V$ are a uniform flow velocity in the streamwise and transverse direction, respectively. The gage pressure is defined as $P$.

| Boundary           | Velocity        | Pressure |
|--------------------|-----------------|----------|
| Inlet              | $U = 1, V = 0$  | -        |
| Outlet             | $\frac{\partial U}{\partial X} = 0, \frac{\partial V}{\partial X} = 0$ | $P = 0$ |
| Upper and Lower    | Frictionless, $\frac{\partial U}{\partial Y} = 0, V = 0$ | -        |
Cylinder surface

The calculations initiated with the fluid flow velocity $V = 0, U = 0$, at rest, then in few time step (constant $\Delta t = 0.05$) the flow velocity is increased smoothly to $V = 0, U = 1$, and used 40 iteration per time step. The monitor convergence criteria in continuity and velocity are set for $B/A = 1-4$ of 0.001 and 0.00001 for $B/A = 5-7$ to monitor convergent against residual at the end of each iteration during iterative sequence same as Lamtharn and Pimsarn [23] and Sohankar et al. [4, 5, 6].

2.3 Numerical method

For this study in solving the unsteady, conservative, continuity and dimensionless form of the 2-D incompressible flow Navier–Stokes equations with a constant fluid property are given as follows.

Continuity:

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0$$

(1)

Momentum-x:

$$\frac{\partial U}{\partial T} + \frac{\partial (UU)}{\partial X} + \frac{\partial (UV)}{\partial Y} = - \frac{\partial P}{\partial X} + \frac{1}{Re} \left( \frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2} \right)$$

(2)

Momentum-y:

$$\frac{\partial V}{\partial T} + \frac{\partial (UV)}{\partial X} + \frac{\partial (VV)}{\partial Y} = - \frac{\partial P}{\partial Y} + \frac{1}{Re} \left( \frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2} \right)$$

(3)

With

$$U = \frac{u_x}{u_\infty}, V = \frac{u_y}{u_\infty}, T = \frac{t u_\infty}{d}, X = \frac{x}{a}, Y = \frac{y}{d}, P = \frac{p}{\rho u_\infty^2}$$

(4)

The Reynolds number is defined as $Re = \frac{U_\infty d}{v}$, where $v$ is kinematic viscosity, $d = B \sin \alpha$, $(90-\beta)^\circ < \alpha \leq 90^\circ$ and $0^\circ \leq \beta \leq 60^\circ$ and $d = A \cos (\alpha+\beta)/\cos \beta + B \sin \alpha$, $0^\circ \leq \alpha \leq (90-\beta)^\circ$.

The Direct Numerical Simulation (DNS) with a finite volume method, non-uniform meshing with second-order implicit time discretization into eight-node quadratic quadrilateral elements are employed. An incompressible flow SIMPLEC code with constant fluid properties such as kinematic viscosity and density are used. The convective terms using a third-order QUICK scheme same as [4, 5, 6, 23]. The same numerical approach as mentioned above has been successfully adopted in the work of [23] for the simulations of flow past over a single square and right trapezoidal cylinder is used in the present study.

3. Results and discussion

**Benchmark numerical model**

**Table 2.** Comparison of results at $Re = 100$ for a fixed square cylinder subjected flow past around.

| Works by | CL RMS | $St$ | Percentage Difference (%) |
|----------|--------|------|---------------------------|
| Darekar and Sherwin [14] (3D, simulation) | 0.1860 | 0.1460 | 1.1 0.7 |
| Zhao et al. [7] (2D, simulation) | 0.1908 | 0.1447 | 1.5 0.2 |
| Sahu et al. [10] (2D, simulation) | 0.1880 | 0.1486 | 0.0 2.5 |
| Sen et al. [11] (2D, simulation) | 0.1928 | 0.1452 | 2.5 0.2 |
| Sharma and Eswaran [8] (2D, simulation) | 0.1922 | 0.1488 | 2.2 2.7 |
| Singh et al. [9] (2D, simulation) | 0.1600 | 0.1470 | 14.9 1.4 |
| Sohankar et al. [6] (2D, simulation) | 0.1560 | 0.1460 | 17.0 0.7 |
| Okajima [16] (experimental) | - | 0.1400 | - 3.4 |
| Present | 0.1880 | 0.1449 | - |
In order to avoid inaccuracy from a numerical simulation. The benchmark numerical model for a fixed square cylinder subjected flow past around \((B/A = 1, \beta = 0^\circ, \alpha = 0^\circ)\) at \(Re = 100\) is performed and the computational results were validated against the published data in literature as presented in Table 2. By using the RMS lift coefficients and the Strouhal number as an indicator the strong sensitivity \([6, 7, 23]\) represent to another sensitivity numerical parameters from fully at least ten cycles saturated flow were calculated. These results indicated that the results of the RMS lift coefficients values predicted from in this study very good agreement with the published numerical results in Zhao et al. \([7]\), Sharma and Eswaran \([8]\), Sahu et al. \([10]\), Sen et al. \([11]\) and Darekar and Sherwin \([14]\). The percentage difference between published data works with the present study of these RMS lift coefficients is less than 3%. The numerical results of Strouhal number, \(St\) from different models are very close to each other. The percentage difference between numerically published works with the present study of these Strouhal number is less than 3% and about 3.4% with experimental works of Okajima \([16]\).

### 3.1 Effect of sharpening angle and side ratio on RMS lift coefficients

A comprehensive numerically investigate of the influence of the sharpening angle from \(0^\circ, 15^\circ, 22.5^\circ, 40^\circ\) and \(60^\circ\) with side ratio \(B/A = 1, 2, 3, 4, 5, 6\) and \(7\) were studied for various quantities parameters such as flow force coefficients and vortex shedding frequency responses. In figure 3 (left top) shown the variations of RMS lift coefficients with sharpening angle for side ratio \(B/A = 1, 2, 3, 4, 5, 6\) and \(7\) and (left bottom) shown that with side ratio for sharpening angle \(\beta = 0^\circ, 15^\circ, 22.5^\circ, 40^\circ\) and \(60^\circ\). The RMS lift coefficients abruptly at around \(B/A \geq 3\) monotonic increase as sharpening angle increases, in contrast, the RMS lift coefficients abruptly at around \(\beta \geq 40^\circ\) monotonic decrease as side ratio increases. For instance, the RMS lift coefficients at \(B/A = 4\) is lower than about 4.4 times at \(B/A = 1\) for \(\beta = 0^\circ\) and lower than about 2 times at same side ratio for \(\beta = 15^\circ, 22.5^\circ, 40^\circ\). For \(\beta = 0^\circ\) that is a square cylinder for \(B/A = 1\) and rectangular cylinder for \(B/A = 2, 3, 4, 5, 6\) and \(7\), the RMS lift coefficients slightly monotonic decrease from the square section to longer rectangular section.

**Figure 3.** (left top) The \(CL_{RMS}\) as a function of sharpening angle, (left bottom) the \(CL_{RMS}\) as a function of side ratio, (right-top) The Strouhal number against the sharpening angle , (right bottom) the Strouhal number against the side ratio.
3.2 Effect of sharpening angle and side ratio on Strouhal number

In figure 3 (right-top) shown the variations of Strouhal number with sharpening angle for side ratio \( B/A = 1, 2, 3, 4, 5, 6 \) and \( 7 \) and (right bottom) shown that with side ratio for sharpening angle \( \beta = 0^\circ, 15^\circ, 22.5^\circ, 40^\circ \) and \( 60^\circ \). The Strouhal number slightly change with smoothly monotonic increase as sharpening angle increases, in contrast, the Strouhal number slightly change with smoothly monotonic decrease as side ratio increases until at \( B/A = 4 \) to \( 5 \), the Strouhal number abruptly decrease. It then Strouhal number slightly decrease again for larger side ratio as shown in figure 3 (right bottom). For instance, the Strouhal number at each \( B/A = 2, 3, 4, 5, 6 \) and \( 7 \) is lower than at \( B/A = 1 \) with some constant value for same sharpening angle. For \( \beta = 0^\circ \) that is square cylinder for \( B/A = 1 \) and rectangular cylinder for \( B/A = 2, 3, 4, 5, 6 \) and \( 7 \), the Strouhal number slightly monotonic decrease from the square section to longer rectangular section.

4. Conclusion

The simulation of a single right trapezoidal cylinder immersed in a uniform oncoming unsteady laminar flow regime with low Reynolds number, \( Re = 100 \), and with various sharpening angles in the range of \( 0^\circ, 15^\circ, 22.5^\circ, 40^\circ \) and \( 60^\circ \) accompany with the cylinder side ratio in the range of \( 1, 2, 3, 4, 5, 6 \) and \( 7 \) have been performed.

The increasing of sharpening angle with constant side ratio, the Strouhal number slightly increases whilst the root mean square lift coefficients significantly increase. As the side ratio increases with constant sharpening angle, the Strouhal number slightly decreases and the RMS lift coefficients evidently decrease. The larger side ratio is lesser flow force than smaller side ratio for the single right trapezoidal cylinder. The Strouhal number and the RMS lift coefficients in this study are provided to additional data for use in the absence of more applicable data for engineering design of the single right trapezoidal cylinder in structural integrity point of view further.

References

[1] Williamson C H K 1996 Annu. Rev. Fluid Mech. 28 477–539
[2] Zdravkovich M M 1997 Flow around Circular Cylinders (New York: Oxford University Press)
[3] Prasanth T Behara S Singh S Kumar R Mittal S 2006 J. Fluids and Structures 22 865–876
[4] Sohankar A Davison L Norberg C 1995 Proc. Twelfth Australasian Fluid Mech. Conf. 517-520
[5] Sohankar A Norberg C Davison L 1997 J. Wind Eng. Ind. Aerodyn 69-71 189–201
[6] Sohankar A Norberg C Davison L 1998 Int. J. Numer. Meth. Fluids 26 39–56
[7] Zhao M Cheng L Zhou T 2013 Phys. Fluids 25 023603-25
[8] Sharma A Eswaran V 2004 Numer. Heat Transfer, Part A 45 247–269
[9] Singh A P De A K Carpenter V K Eswaran V Muralidhar K 2009 Int. J. Numer. Methods Fluids 61 658–682
[10] Sahu A K Chhabra R P Eswaran V 2009 J. Non-Newtonian Fluid Mech. 160 157–167
[11] Sen S Mittal S BiswasG 2011 Int. J. Numer. Methods Fluids 67 1160–1174
[12] Sohankar A Norberg C Davidson L 1999 Phys. Fluids 11 288–306
[13] Robichaux J Balachandar S Vanka S P 1999 Phys. Fluids 11 560–578
[14] Darekar R M Sherwin S J 2001 J. Fluid Mech. 426 263–295
[15] Saha A K Biswas G Muralidhar K 2003 Int. J. Heat Fluid Flow 24 54–66
[16] Okajima A 1982 J. Fluid Mech. 123 379–398
[17] Okajima A 1990 J. Wind Eng. Ind. Aerodyn. 33 171–180
[18] Okajima A Yi D Sakuda A Nakano T 1997 J. Wind Eng. Ind. Aerodyn. 67–68 91–102
[19] Arnab A K Dalal A 2006 Int. J. Numer. Meth. Fluids 52 801–821
[20] Alawadhi E M 2013 J. Fluids Eng. 135 041202–10
[21] Jiang H Cheng L 2018 Phys. Fluids 30 104107–13
[22] Chung Y J Kang S H 2000 Phys. Fluids 12 1251–1254
[23] Lamtharn S Pimsarn M 2018 MATEC Web of Conferences ICEAST 2018 192 02030