International Journal of Engineering Trends and Technology
ISSN: 2231–5381 / https://doi.org/10.14445/22315381/IJETT-V71I5P227
Volume 71 Issue 5, 259-265, May 2023

Original Article

Numerical Analysis of Incompressible Low-Re Impulse-Flow Over Staggered 2D Circular Cylinders

Yagneshkumar A Joshi[1], Ramesh Bhoraniya[2], A B Harichandan[3]

[1,2]*Department of Mechanical Engineering, Marwadi University, Rajkot, India.*
[3]*Department of Aerospace Engineering, KIIT Bhubneswar, India.*

[1]*Corresponding Author : yagneshkumar.joshi101470@marwadiuniversity.ac.in*

Received: 03 January 2023 Revised: 31 March 2023 Accepted: 12 May 2023 Published: 25 May 2023

Abstract *- Cylinders are essential in constructing several engineering designs where fluid passes over them. Under varying environmental conditions, the fluid flows with impulse, affecting design structures considerably. This paper presents the numerical analysis of incompressible low-Re gusty flow with impulse inlet passing through three staggered cylinders. The problem is numerically simulated and investigated for two distinct variable factors keeping one constant at a time: inlet gust frequency and Reynolds number. The vorticity contours are obtained that provide information about local rotation. The streamlines are obtained to identify the wake region. The C_L, C_D, and St are computed. The results show a beneficial effect of variability in impulse intake velocity on the wake region generated between and behind the cylinders. It represents enhanced flow characteristics derived from low angular gust frequency value of intake velocity fluctuation.*

Keywords *- CFRUNS, Incompressible N-S solver, Low-R, Staggered circular cylinders.*

1. Introduction

Researchers are keenly interested in the flow through cylindrical bodies due to its many mechanical, civil, and marine engineering applications. The researchers have investigated it extensively and reported fundamental phenomena associated with it. Some of them are briefly summarized as followings. Braza et al. [1] reported viscous flow over a circular cylindrical bodies with Re values 100, 200, and 1000. He used a method based on velocity-pressure formulation and a convective scheme. D. J. Tritton [3] experimentally measured drag on circular cylindrical bodies in the range of Re 0.5 to 100. It provides information about vortex sheet transition in the wake region. Harichandan and Roy[5] introduced the "CFRUNS" cell flux reconstruction unstructured triangular grid fully explicit scheme solver. They employed it to investigate flow across two distinct configurations of cylinders under uniform fluid flow. Jiro Mizushimaa and Norihisa Suehiro[6] investigated uniform flow through tandem cylinders with a high Reynolds number, focusing on the gap ratio's effect. Papaioannou et al.[8] investigated the impact of 2D and 3D research on tandem cylinder flow. Yeo et al.[9] have conducted a numerical investigation utilizing FDM for uniform flow around cylinders. Lin et al.[11] carried out an investigation of the tandem cylinder layout for uniform flow and evaluated the effect of L/D in-depth, experimentally. Xiaoqiu et al. [14] studied three staggered circular cylinders in a symmetrical arrangement and correlated the wake frequency and interval among the cylinders. Salwa Fezai et al. [16] conducted a simulation-based investigation on two configurations of square cylinders at low-Re. They reported a comparative study of both the configuration and its effect on the lift and drag coefficient of cylinders. They also highlighted the effect of cylinder arrangement on the critical Re value. Haider et al. [18] have simulated different arrangements of circular cylinders under uniform flow with low Re. He observed that the cylinders in a staggered arrangement have better heat transfer characteristics. However, their velocity profile demonstrates considerable wake region behind cylinders. Meneghini.et al. [19] reported flow interface between cylinders in differenct arrangements. The author chirag et al. [21] reported gusty flow interface numerically using CFRUNS scheme for flow over single circular cylinder at different gust frequencies. and explained the effect the gust on separation point. The authors Joshi et al. [22] investigated the side-by-side cylinder arrangements for flow under gusty effect and reported reduced wake. Hence the authors see the scope for improvement in flow characteristics by means of reduction in wake behind the cylinders as a research gap. An attempt is made to numerically investigate the problem with flow under artificial gust effect and to identify the better design conditions for staggered cylinder arrangement to have minimum wake region in specific circumstances.

2. Problem Definition

A flow problem with 2D cylindrical bodies in a staggered triangular arrangement with two different configurations is influenced by incompressible gusty fluid flow with a low Reynolds number (low Re). Here, the gusty flow represents the flow with velocity variation based on the cosine function. The problem is physically described in Fig.1 (a) & (b). The first configuration is with two upstream circular cylinders and a single downstream cylinder, as shown in Fig.1 (a).

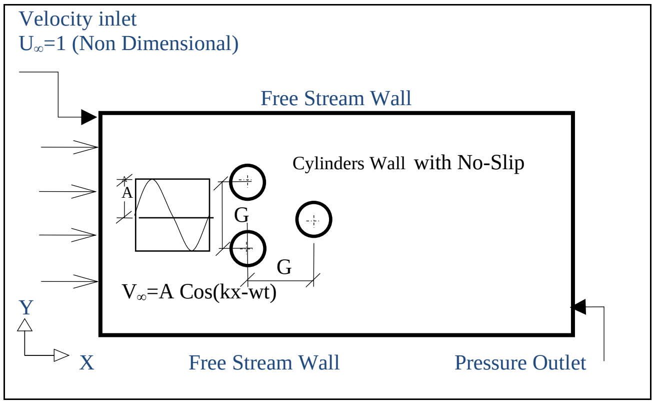


Fig. 1 (a) Physical representation of problem configuration with single cylinder downstream

The second configuration, with one single cylinder on the upstream side and two on the downstream side, is shown in Fig.1 (b).

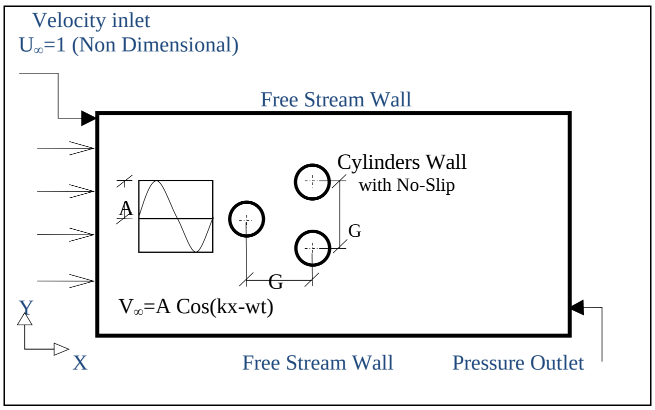


Fig. 1 (b) Physical representation of problem configuration with single cylinder upstream

3. Solution Methodology

As per the above configurations, the problem is simulated using the CFRUNS solver developed by Harichandan and Roy [5]. The solver is modified to incorporate the gusty effect in the flow to create impulses in a specified region of the inlet flow field that acts as a source. A cosine wave source is employed to provide v-velocity variation in the y-direction. Under the combined effect of u-velocity inlet and source, the cylinders experience gusty flow impulses. The effect of flow variations is captured in terms of the physical wake region between and behind the cylinders. The lift force coefficient and drag force coefficients for cylinder walls are calculated. The vorticity contours are obtained that provide information about local rotation. The streamlines are obtained to identify the wake region. The inlet, upper, and lower flow domain limits are 10D from the body's centre. And the outflow limit is set at 25D from the nearest cylinder centre. The outer boundaries are applied with free-stream Dirichlet boundary conditions, and the body surface is applied with no-slip boundary conditions.

3.1. Governing Equations

Three basic equations govern the fluid flow involved in this problem. These are the continuity equation (conservation of mass law), the 2D Navier-Stokes x-momentum equation, and the y-momentum equations (conservation of momentum law), as shown in the differential form below.

Conservation of Mass Equation

$$\nabla . v = 0 \qquad (1)$$

Momentum in X direction

$$\frac{\partial v_x}{\partial t} + \frac{\partial (v_x{}^2)}{\partial x} + \frac{\partial (v_x v_y)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{Re\left(\frac{\partial^2 v_x}{\partial x^2} + \frac{\partial^2 v_x}{\partial y^2}\right)} \qquad (2)$$

Momentum in Y direction

$$\frac{\partial v_y}{\partial t} + \frac{\partial (v_x v_y)}{\partial x} + \frac{\partial (v_y{}^2)}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{Re\left(\frac{\partial^2 v_y}{\partial x^2} + \frac{\partial^2 v_y}{\partial y^2}\right)} \qquad (3)$$

Where,

v_x represents the horizontal velocity component,
v_y represents the vertical velocity component,
p represents the pressure-to-density ratio,
t represents the non-dimensional time
U_∞ represents the Free stream velocity.

The numerical simulation solver code solves governing equations iteratively. First-order accuracy in time and second-order accuracy in space characterize the solver. It follows SIMPLE algorithm steps for a solution where velocity and pressure gradients are initially computed based on boundary conditions. Then it computes the instantaneous velocity fields by solving the discretized momentum equations. Afterwards, it corrects the pressure using the pressure poison equation. The updated pressure is then applied to update boundary pressure conditions. The fluxes and cell velocities are then corrected.

As mentioned above, a source velocity component is employed in an upstream region defined as follows:

$$v_g = A \cos(kx - wt) \qquad (4)$$

Where 'A' is the amplitude of gust intensity reference to the flow, 'k' represents the wave number, 'x' represents the source x-coordinate, and 'w' represents the angular gust frequency.

3.2. Solver Validation

The results of an identical flow problem are compared against numerical and experimental data published by researchers for a benchmark flow issue, "uniform flow past a single cylinder", to validate the code. The time progression study of C_L & C_D for low Re (Re 100) uniform flow across a cylinder is depicted in Figure 2.2. It shows periodic oscillation with respect to time, which represents the shedding of vortices alternately initiating from the cylinder's top and bottom.

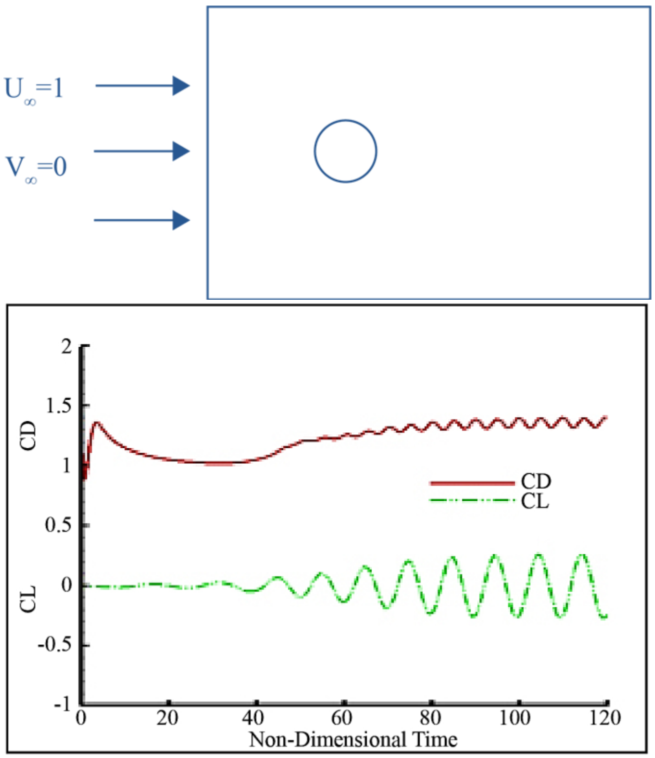


Fig. 2 Model flow problem and its C_L and C_D evolution at Re 100 [13]

In Table 1, computed C_L & C_D values are compared to data released by other researchers for a comparable problem. Observations indicate that the computed findings are comparable. It demonstrates the code's predictability for handling comparable issues with low-Re.

Table 1. Comparison of computed data with published data flow over a cylinder (Re 100)

Author	C_L	C_D
Chassing et al. [1]	1.364 ± 0.02	± 0.25
Tritton [3]	1.320 ± 0.01	NA
A B Harichandan et al. [5]	1.352 ± 0.01	± 0.278
Yeo et al. [9]	1.356 ± 0.01	± 0.287
Saltara et al. [19]	1.370 ± 0.01	NA
Present code Results	1.359 ± 0.01	± 0.251

3.3. Grid Independence

The grid independence is carried out using 80, 120, 160, and 200 line elements on the cylinder's circumference. Referred to Table 2, the results are significantly refined as the number of elements on the cylinder circumference increases from 40 to 160. However, after 160, there is no substantial variation.

Table 2. Result for grid test for single cylinder (Re 100)

Elements on cylinder circumference of cylinder	C_L	C_D	St
80	1.112 ± 0.021	0.181	0.168
120	1.185 ± 0.015	0.210	0.164
160	1.352 ± 0.010	0.278	0.161
200	1.381 ± 0.010	0.281	0.161

4. Results and Discussion

As mentioned in Section 2, two different configurations of cylinders in the triangular arrangement are studied under gusty flow. Figure 3 shows the grid generated for simulation with 160 elements on the cylinder wall. Figure 4 represents comparative streamlined plot results. Figure 5 shows the contour plot for the said two configurations.

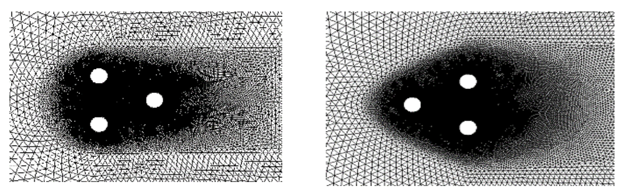


Fig. 3 Grid generation

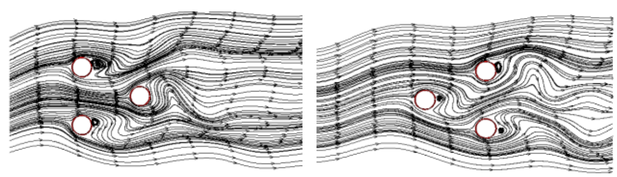


Fig. 4 Streamline plot for Re = 100, T = 3D, and w = 0.25π

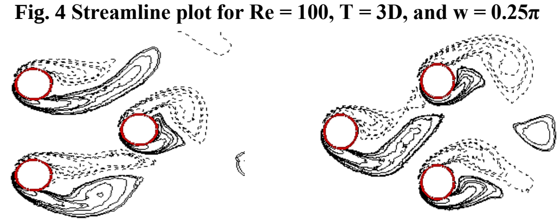


Fig. 5 Vorticity contours plot for Re = 100, T = 3D, and w = 0.25

Figure 6 shows the time evaluation results for the two configurations. It is observed that out of the two stated arrangements, the second one, wherein a single cylinder is upstream, and two cylinders are downstream, shows more fluctuations in C_L and C_D comparatively. That indicates more fluctuating flow in the second configuration compared to the first one.

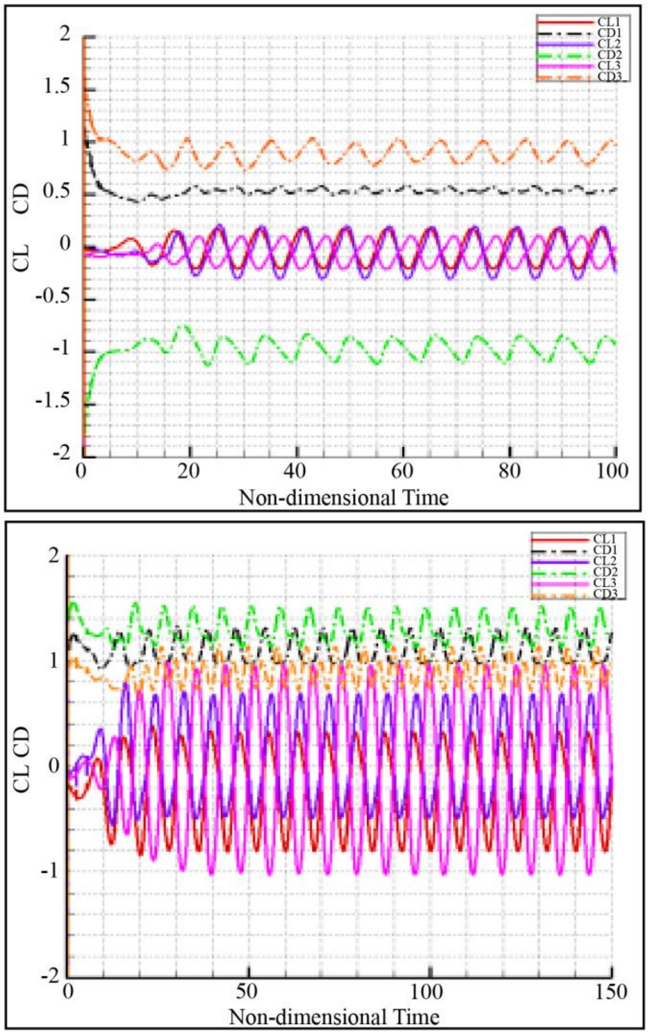
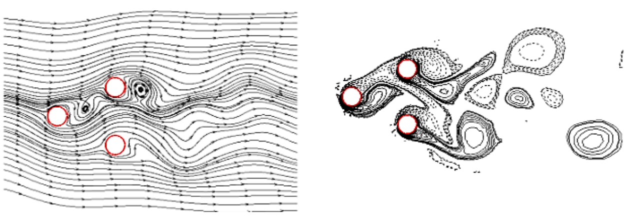
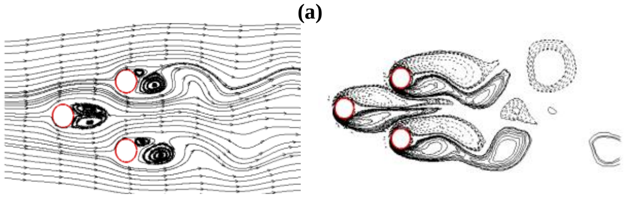


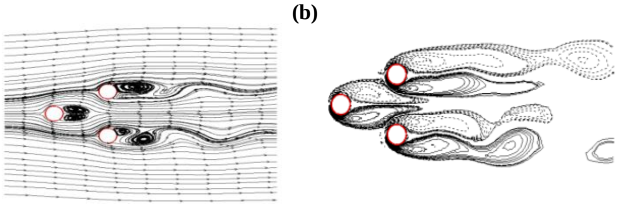
Fig. 6 The time progression study of C_L & C_D for w = 0.25



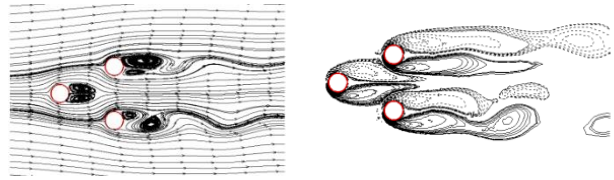
(a)



(b)



(c)



(d)

Fig. 7 Streamline plot for Re = 100, T = 3D, and (a) w = 0.25π, (b) w =0.50π, (c) w = 0.75π, (d) w = 1.0π

Further, considering more fluctuating characteristics of the second type of arrangement, it is studied with four gust frequency values at Re 100 to study wake characteristics in this case. The gust frequencies employed for the study are (a) w = 0.25π, (b) w =0.50π, (c) w = 0.75π, (d) w = 1.0π.

Figure 7 represents the comparative result plots of streamline and velocity contour that give insight into wake at different gust frequency values.

As visible in Figure 7, for the considered value of Re, with low gust frequency, i.e.0.25π, the amount of wake generated is minimum. With increased gust frequency to 1.0π, flow behaves like uniform flow, and sows increased wake region. It shows that an artificial low gust frequency flow can help to reduce the amount of wake in a low Reynolds number flow.

Table 3. Result for flow over the staggered cylinder (Re 100)

Parameters		C_L	C_D
		Re = 100	Re = 100
W=0.25π	UC	0.5483 ± 0.0441	-0.016 ± 0.1905
W=0.50π	UC	0.39965 ± 0.03295	-0.0166 ± 0.0181
W=0.75π	UC	0.3961 ± 0.0303	-0.0158 ± 0.0049
W=1.0π	UC	0.389 ± 0.0327	-0.016 ± 0.0027
W=0.25π	DUC	-0.97655 ± 0.13515	-0.0531 ± 0.2519
W=0.50π	DUC	-0.93335 ± 0.02675	-0.0328 ± 0.0413
W=0.75π	DUC	-0.9351 ± 0.0323	-0.0304 ± 0.0275
W=1.0π	DUC	-0.9492 ± 0.0429	-0.0255 ± 0.0227
W=0.25π	DLC	0.90975 ± 0.11755	-0.0538 ± 0.1598
W=0.50π	DLC	0.80765 ± 0.04535	-0.0536 ± 0.0151
W=0.75π	DLC	0.80015 ± 0.04245	-0.0518 ± 0.0117
W=1.0π	DLC	0.78535 ± 0.04085	-0.0538 ± 0.0107

During the study, it is found that, for the same gust frequency, with the increase in Reynolds number from 50 to 75 to 100, there is no significant variation in the flow pattern except the elongation of flow contours downstream side of the cylinders. It supports the theoretical argument that the flow will travel fast at increased Re due to increased velocity, resulting in an elongated pressure region.

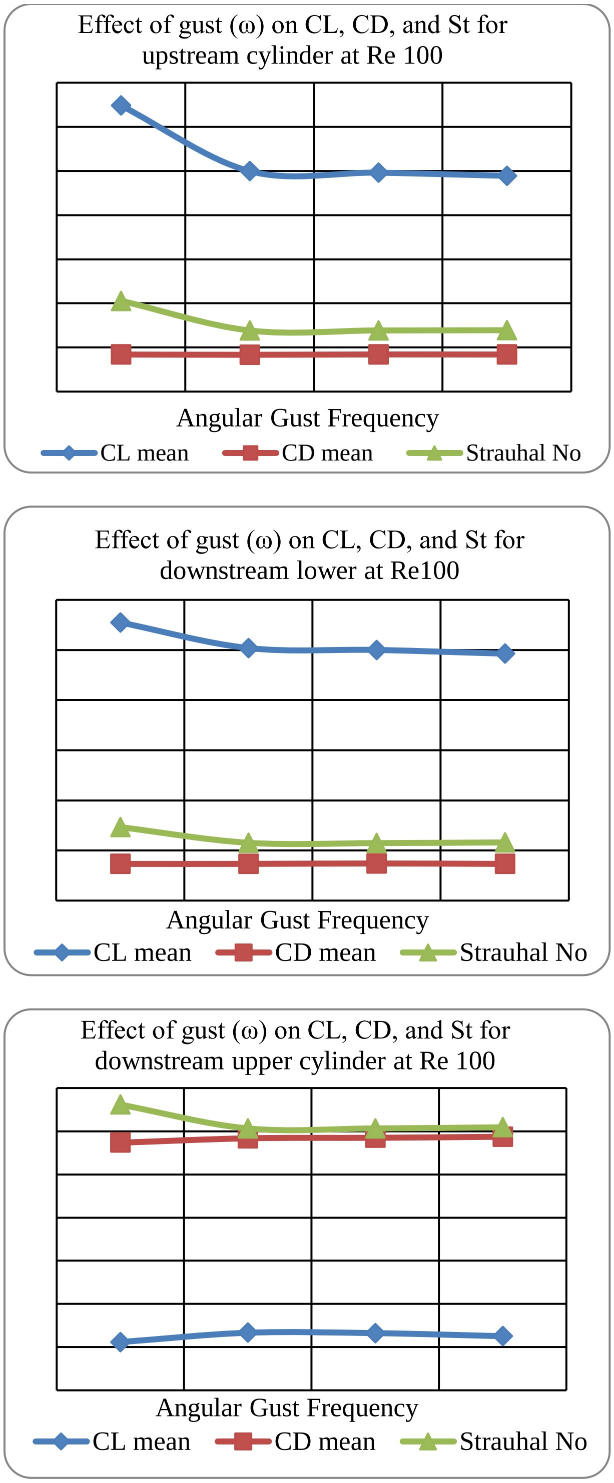


Fig. 8 Effect of gust (ω) on C_L, C_D, and S_t at Re 100

Table 3 shows the results for the effect of gust frequency on the upstream single cylinder, downstream upper cylinder, and downstream lower cylinder.

The results for Re 100 that are listed in Table 3 are represented in fig.8 in terms of charts, comparing the effect of gust frequency variation on the C_L, C_D and St. The analysis shows that there is a significant difference in the effect of gust frequency variation on a single upstream cylinder, downstream upper cylinder, and downstream lower cylinder as shown in three different charts in fig.8.

Similarly, as the study carried out and discussed above, the analysis is also done for the other two values of Reynolds number for Re 75 and Re 50, as mentioned previously. The results are listed in Table 4 and Table 5.

Table 4. Result for flow over the staggered cylinder (Re 75)

Parameters		C_L	C_D
		Re = 75	**Re = 75**
W=0.25π	UC	0.50425 ± 0.03525	-0.0153 ± 0.1668
W=0.50π	UC	0.37235 ± 0.02565	-0.01505 ± 0.01625
W=0.75π	UC	0.3651 ± 0.029	-0.0138 ± 0.0039
W=1.0π	UC	0.36505 ± 0.02835	-0.0138 ± 0.0023
W=0.25π	DUC	-0.96185 ± 0.14985	-0.05025 ± 0.21265
W=0.50π	DUC	-0.863 ± 0.0165	-0.0345 ± 0.0227
W=0.75π	DUC	-0.85855 ± 0.02355	-0.0348 ± 0.0125
W=1.0π	DUC	-0.86575 ± 0.02455	-0.03415 ± 0.00965
W=0.25π	DLC	0.8686 ± 0.1528	-0.04865 ± 0.13225
W=0.50π	DLC	0.79695 ± 0.04025	-0.0472 ± 0.01
W=0.75π	DLC	0.78935 ± 0.02535	-0.0486 ± 0.0037
W=1.0π	DLC	0.7879 ± 0.0245	-0.04945 ± 0.00215

Table 5. Result for flow over the staggered cylinder (Re 50)

Parameters		C_L	C_D
		Re = 50	**Re = 50**
W=0.25π	UC	0.42905 ± 0.04845	-0.0161 ± 0.141
W=0.50π	UC	0.35445 ± 0.03235	-0.01285 ± 0.02345
W=0.75π	UC	0.3456 ± 0.0294	-0.00888 ± 0.00792
W=1.0π	UC	0.3467 ± 0.0263	-0.01325 ± 0.00235
W=0.25π	DUC	-0.8975 ± 0.1234	-0.0601 ± 0.1498
W=0.50π	DUC	-0.83845 ± 0.02055	-0.0481 ± 0.0127
W=0.75π	DUC	-0.8414 ± 0.0294	-0.0477 ± 0.0029
W=1.0π	DUC	-0.8446 ± 0.0207	-0.04805 ± 0.00205
W=0.25π	DLC	0.84335 ± 0.12785	-0.05135 ± 0.09695
W=0.50π	DLC	0.7951 ± 0.0382	-0.0502 ± 0.0078
W=0.75π	DLC	0.801 ± 0.0264	-0.05065 ± 0.00185
W=1.0π	DLC	0.8059 ± 0.0226	-0.0513 ± 0.0015

Considering all the above investigations and results, it is observed that the flow approaches uniform flow as the values of gust frequency increase within range. The amount of the wake behind the cylinders is reduced considerably as the values of gust frequency decrease within the range. That is because; the cosine function impulse creates continuous variation in the flow direction and creates a gusty effect in the x direction. That is providing a shift of flow separation points continuously. This variation does not allow the wake to settle down in the region behind the cylinder, specifically at low gust frequencies. And considerably reduced wake in the region behind the cylinders is observed.

5. Conclusion

The extension of fully explicit code with CFR scheme for gusty flow can capture the flow patterns with significantly less computational facility, provided that it takes considerable time to solve the flow problem. As a key outcome of the work, the results demonstrate the positive effect of low gust frequency impulse (i.e. 0.25π) in the incompressible flow in terms of narrowing down the wake region behind the cylinders. It is also to be noted that the values of C_L and C_D with gusty impulse flow are a little lower than that of uniform flow with the same Re.

Abbreviations

USC	Upstream Single Cylinder
DUC	Downstream Upper Cylinder
DLC	Downstream Lower Cylinder
DSC	Downstream Single Cylinder
UC	Upper Cylinder
Low-Re	Low Reynolds Number
C_D	Drag force coefficient
C_L	Lift force coefficient
S_t	Srouhal Number
W	Angular Gust Frequency

Acknowledgments

The authors acknowledge the effort of Dr A Roy and Dr A B Harichandan to develop a CFR scheme and explicit solver for the solution of uniform incompressible flow. The author also acknowledges the support of all who have directly or indirectly motivated the work.

References

[1] M. Braza, P. Chassaing, and H. Ha Minh, “Numerical Study and Physical Analysis of the Pressure and Velocity Fields in the Near Wake of A Circular Cylinder,” *Journal of Fluid Mechanics*, vol. 165, no. 1986, pp. 79–130, 1986. [Crossref] [Google Scholar] [Publisher Link]

[2] Amir H. Ghahremani, and Reza Saleh, "Numerical Study of Turbulence Models in Heat Transfer of a Confined and Submerged Jet Impingement Using AL2O3 - Water Nano Fluid," *SSRG International Journal of Mechanical Engineering*, vol. 2, no. 5, pp. 47-55, 2015. [Crossref] [Google Scholar] [Publisher Link]

[3] D. J. Tritton, “Experiments on the Flow Past a Circular Cylinder at Low Reynolds Numbers,” *Journal of Fluid Mechanics*, vol. 6, no. 4, pp. 547–567, 1959. [Crossref] [Google Scholar] [Publisher Link]

[4] B.N.C. Mohan Reddy, J.A.Sandeepkumar, and A.V. Hari Babu, "Performance Analysis of Alternative Refrigents Inside Capillary Tube of Refrigeration System," *SSRG International Journal of Mechanical Engineering*, vol. 4, no. 12, pp. 8-23, 2017. [Crossref] [Publisher Link]

[5] Atal Bihari Harichandan, and Arnab Roy, “Numerical Investigation of Low Reynolds Number Flow Past Two and Three Circular Cylinders Using Unstructured Grid CFR Scheme,” *International Journal of Heat and Fluid Flow*, vol. 31, no. 2, pp. 154–171, 2010. [Crossref] [Google Scholar] [Publisher Link]

[6] Jiro Mizushima, and Norihisa Suehiro, “Instability and Transition of Flow Past Two Tandem Circular Cylinders,” *Physics of Fluids*, vol. 17, no. 10, 2005. [Crossref] [Google Scholar] [Publisher Link]

[7] Amirreza Mohammadian, Suresh Akella, and A.M.K Prasad, "Direct Numerical Simulation of Turbulent Flow Around an Impulsively Started Circular Cylinder by Using Mesh-Free Vortex Method," *SSRG International Journal of Mechanical Engineering*, vol. 2, no. 11, pp. 13-18, 2015. [Crossref] [Publisher Link]

[8] Georgios V. Papaioannou et al., “Three-Dimensionality Effects in Flow Around Two Tandem Cylinders,” *Journal of Fluid Mechanics*, vol. 558, pp. 387–413, 2006. [Crossref] [Google Scholar] [Publisher Link]

[9] H. Ding et al., “Numerical Simulation of flows Around Two Circular Cylinders by Mesh-Free Least Square-Based finite Difference Methods,” *International Journal for Numerical Methods in Fluids*, vol. 53, no. 2, pp. 305–332, 2007. [Crossref] [Google Scholar] [Publisher Link]

[10] Nidhul K, Sunil A S, and Kishore V, "Influence of Lateral Boundaries and Grid Spacing on Steady Flow Past a Square Cylinder," *SSRG International Journal of Mechanical Engineering*, vol. 1, no. 8, pp. 9-12, 2014. [Crossref] [Publisher Link]

[11] J. C. Lin, Y. Yang, and D. Rockwell, “Flow Past Two Cylinders in Tandem: Instantaneous and Averaged Flow Structure,” *Journal of Fluids and Structures*, vol. 16, no. 8, pp. 1059–1071, 2002. [Crossref] [Google Scholar] [Publisher Link]

[12] D.Sarath Chandra et al., "Impact of Different Volume Concentrations and Flow Rates on the Thermal Performance of Counter Flow Cylindrical Shell and Helical Coil Heat Exchanger Using Cu/H2O Nano Fluids," *SSRG International Journal of Thermal Engineering*, vol. 6, no. 3, pp. 11-15, 2020. [Crossref] [Publisher Link]

[13] S. Nallusamy, "Thermal Conductivity Analysis and Characterization of Copper Oxide Nanofluids through Different Techniques," *Journal of Nano Research*, vol. 40, pp. 102–112, 2015. [Crossref] [Google Scholar] [Publisher Link]

[14] Xiaoqiu HE, Yongliang Xiong, and Dan Yang, “Numerical Investigation on the Flow Past Three Cylinders in Staggered Symmetrical Arrangement,” *2018 IEEE 8th International Conference on Underwater System Technology: Theory and Applications (USYS)*, pp. 1–5, 2018. [Crossref] [Google Scholar] [Publisher Link]

[15] Manideep Roy, and Sagar Debnath, "Simulation of Single-Phase Flow Past Cylinder," *SSRG International Journal of Mechanical Engineering*, vol. 7, no. 2, pp. 33-36, 2020. [Crossref] [Publisher Link]

[16] Salwa Fezai, Racha Nefzi, and Brahim Ben-Beya, “Analysis of the Interaction in Flow Around Three Staggered Square Cylinders at Two Different Triangular Arrangements,” *Global Journal of Engineering Sciences*, pp. 1–13, 2021. [Crossref] [Publisher Link]

[17] Y.J. Wagh, and V.H.Patil, "Design and Optimization of Flow Field of Tangentially Fired Boiler," *SSRG International Journal of Mechanical Engineering,* vol. 1, no. 4, pp. 1-7, 2014. [Crossref] [Google Scholar] [Publisher Link]

[18] M. J. Haider et al., “Heat Transfer and Fluid Flow Over Circular Cyclinders in Cross Flow,” *NUST Journal of Engineering Science,* vol. 3, pp. 67–77, 2010. [Google Scholar]

[19] J.R. Meneghini, “Numerical Simulation of Flow Interference Between Two Circular Cylinders in Tandem and Side-By-Side Arrangements,” *Journal of Fluids and Structures*, vol. 15, pp. 327–350, 2001. [Crossref] [Google Scholar] [Publisher Link]

[20] Nishant Kumar, Saurav Upadhaya, and Ashish Rohilla, "Evaluation of the Turbulence Models for the Simulation of the Flow Over a Tsentralniy Aerogidrodinamicheskey Institut (Tsagi)-12% Airfoil," *SSRG International Journal of Mechanical Engineering*, vol. 4, no. 1, pp. 18-28, 2017. [Crossref] [Google Scholar] [Publisher Link]

[21] Chirag Parekh, Arnab Roy, and A. B. Harichandan, “Numerical Simulation of Incompressible Gusty Flow Past a Circular Cylinder,” *Alexandria Engineering Journal,* vol. 57, pp. 3321–3332, 2018. [Crossref] [Google Scholar] [Publisher Link]

[22] Yagneshkumar A. Joshi et al., “Numerical Simulation of Low Reynolds Number Gusty Flow Past Two Side-By-Side Circular Cylinders,” *Proceedings ICOTF KIIT Thermo 2020, Lecture Notes in Mechanical Engineering: Springer*, pp. 71–78, 2020. [Crossref] [Google Scholar] [Publisher Link]