

Simulation of flow past an oscillating triangular cylinder using Finite Element Method

Sutthiphong Srigrarom

Assistant Professor

*Nanyang Technological University, School of Mechanical and Production Engineering,
50, Nanyang Avenue, Singapore, 639798, e-mail: mssrigrarom@ntu.edu.sg*

Abstract

This paper is related to the characteristics, the development and the simulation of a particular fluid-structure interaction phenomenon –the continuous oscillation of an equilateral triangular cylinder in the uniform incoming flow. We propose the explanation of this oscillation. Preliminary analysis indicates that the cylinder could oscillate incessantly by initial positional perturbation or incoming flow fluctuation. It is the unbalance force acting on the cylinder's side faces that causes such movement. On one side, the flow will be flow-past-flat-plate like, whereas the other side will be flow-past-sharp-edge like. Due to the unbalanced pressure exerting on the two sides, the cylinder rotates. When the cylinder moves, these mechanisms switch side interchangeably, and bring the cylinder to continuous oscillation.

Keyword: *fluid-structure interaction, oscillation, triangular cylinder, vortex shedding, Strouhal number, Finite-element method*

Introduction

To date, this coupled fluid-structure interaction has not yet been well studied nor simulated. This is due to the unsteadiness surrounding flow, as well as, continuously moving boundary (the cylinder either oscillates or rotates) around the triangle. The available prior research works were only dealt with the stationary cylinder at different position (Nakashima, 1992 [*ref. 1*]), or with other geometry (Hu et al, 2002 [*ref. 2*], Sakamoto et al, 2001 [*ref. 3*], Naudascher & Wang, 1993 [*ref. 4*], Nakamura & Nakashima, 1993 [*ref. 5*]).

Prior experiments (Srigrarom, 1998 [*ref. 6*]) had been conducted to investigate such behaviour by means of Food coloring dye and Laser Induced Fluorescence flow visualization. The oscillating frequencies appeared to be governed by Strouhal numbers, which fell within limited range of $0.13 < Str < 0.18$. Beyond this range, the cylinder was either stationary or rotates only in one direction

Due to measurement difficulties in experimental study of the unsteady flow pattern in the water tunnel, only qualitative results were obtained. Alternatively, computational fluid dynamics (CFD) simulation presents a viable approach. Recently, the quasi-steady two-dimensional flow study using CFD was conducted recently to get quantitative results and presented in this paper. The uniform incoming flow was implemented for CFD simulation.

Proposed oscillation model

Recently, the author has come up with the explanation of the self-excited oscillation behaviour (Srigrarom, 2003 [*ref. 7*]) In the following figure 1, we consider the flow pattern around a symmetrical cylinder at various positions. The upstream flow is uniform with zero angle of attack, and is brought to stagnation at the front of the cylinder. The flow is divided into two (2) identical streams, upper and lower streams. The divided stream varies smoothly from a 90-degree turn at *S* to a stream merging with the freestream at edge *A* or *B*. At *A* or *B*, the flow separates from the cylinder at both tips or shape edges of the cylinder, creating back flow or eddies on both lateral sides. At the trailing edge *C*, the flow pattern changes according to the freestream Reynolds' number. At low Reynolds' number,

Report Documentation Page				Form Approved OMB No. 0704-0188	
Public reporting burden for the collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to a penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number.					
1. REPORT DATE 14 APR 2005		2. REPORT TYPE N/A		3. DATES COVERED -	
4. TITLE AND SUBTITLE Simulation of flow past an oscillating triangular cylinder using Finite Element Method				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S)				5d. PROJECT NUMBER	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Nanyang Technological University, School of Mechanical and Production Engineering, 50, Nanyang Avenue, Singapore, 639798				8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)				10. SPONSOR/MONITOR'S ACRONYM(S)	
				11. SPONSOR/MONITOR'S REPORT NUMBER(S)	
12. DISTRIBUTION/AVAILABILITY STATEMENT Approved for public release, distribution unlimited					
13. SUPPLEMENTARY NOTES See also ADM001800, Asian Computational Fluid Dynamics Conference (5th) Held in Busan, Korea on October 27-30, 2003. , The original document contains color images.					
14. ABSTRACT					
15. SUBJECT TERMS					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT UU	18. NUMBER OF PAGES 8	19a. NAME OF RESPONSIBLE PERSON
a. REPORT unclassified	b. ABSTRACT unclassified	c. THIS PAGE unclassified			

Simulation of flow past an oscillating triangular cylinder using Finite Element Method

the two divided streams create alternating vortices, shed downstream. At higher Reynolds' number, the two streams join together at alternating streamwise locations creating a turbulent wake downstream of the cylinder.

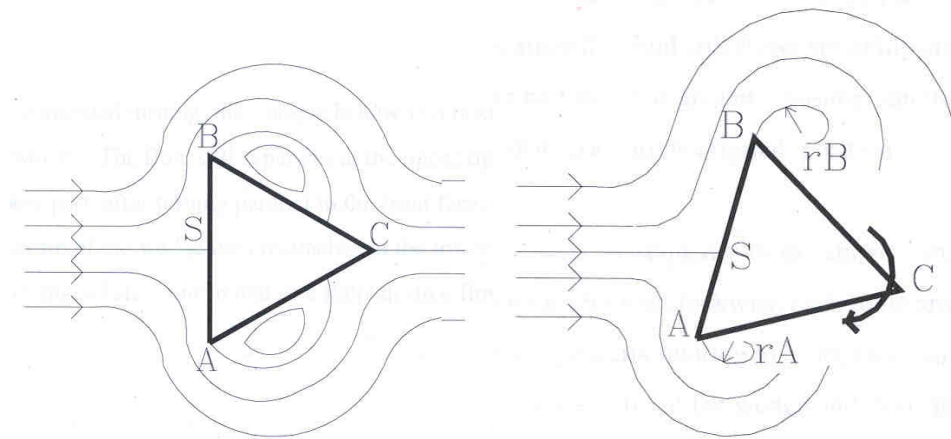


Fig. 1 (left): Flow past the equilateral triangular cylinder at zero degree angle of attack

Fig. 2 (right): Flow past the equilateral triangular cylinder at positive angle of attack

In figures 2 and 3, supposed that the cylinder's symmetry with respect to the freestream is changed slightly. The front face of the cylinder AB inclines, making an angle with the freestream. The flow still separates at both A and B , but the pattern is asymmetrical. The radius of curvature of the streamline separated at A , r_A , is smaller than the radius of curvature at B , r_B . As a result of conservation of angular momentum, the velocity at A is higher than at B ; therefore, the pressure at A is lower than at B ($P_A < P_B$). The cylinder rotates clockwise about the pivot, and the frontal surface AB becomes more inclined to the freestream.

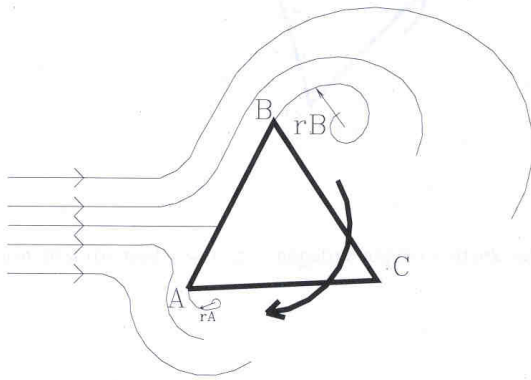


Fig. 3: Flow past the equilateral triangular cylinder at positive angle of attack, schematic diagram (continued).

Due to the continuing turning, the triangle is now in a position where AC is parallel to the freestream. The flow still separates at the upper tip of the cylinder (B), but the flow in the lower part, after turning parallel to the front face, separates only slightly at the lower tip of the cylinder (A), then reattaches to the lower lateral face (AC). The lower flow, then, changes behaviour to that of a flat-plate-like flow.

By virtue of the obvious difference between the two flow patterns, the local pressures differ at the upper and lower parts of the cylinder. The upper part, with the existence of a large eddy, has lower pressure, especially at the core of the eddy, compared with the freestream; whereas at the lower part the pressure is equal to the freestream pressure, since the fluid still flows smoothly and parallel there. Therefore, the flow in the lower part now has greater pressure than the upper part, ($P_A > P_B$) and the cylinder tends to rotate back to its original position.

As a consequence of the above step, the unbalanced pressure forces the cylinder to rotate counterclockwise back to its original position (under the assumption that the cylinder starts rotating in a clockwise direction, as described in the previous step). Due to the inertia of the cylinder and flow, the motion of the cylinder does not stop when it returns to the original position of figure 1. Instead, the cylinder continues to swing in the counterclockwise rotation. As a result, the flow pattern becomes as shown in figures 4 and 5, which correspond to those of figures 2 and 3 flipped upward.

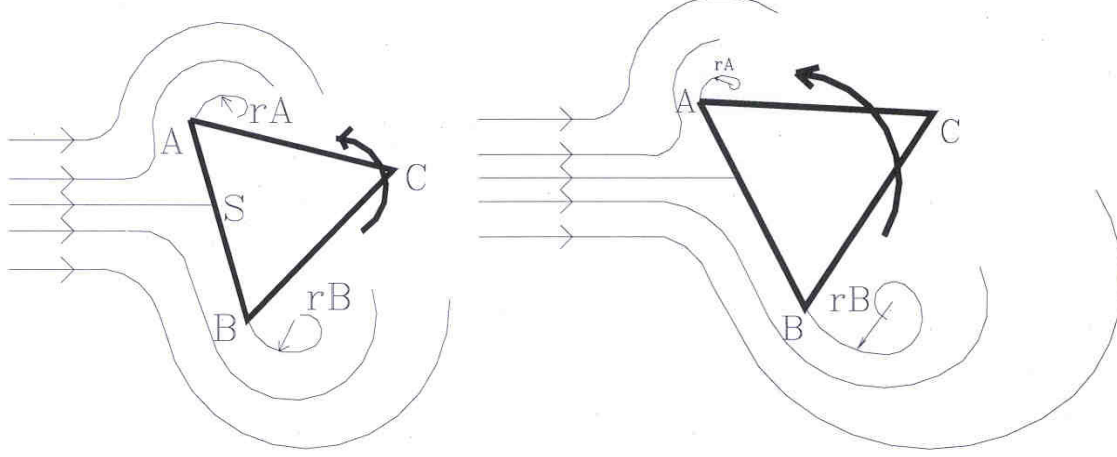


Fig. 4 (left): Flow past the equilateral triangular cylinder at negative angle of attack.

Fig. 5 (right): Flow past the equilateral triangular cylinder at negative angle of attack (continued).

The overall phenomenon can be viewed as the interchange of the flow patterns, from the flow past the sharp edge to flow over the flat plate, and vice versa.

Computational Fluid Dynamics software: FEMLAB

In general, the fluid dynamics problems can be described by Navier-Stokes (NS) equations. There are a lot of software programs available in which NS equations are solved computationally (Computational Fluid Dynamics –CFD). Most of them are developed in order to get the simulation results in a short time with reasonable accuracy. The disadvantages are that they are not very flexible when the user wants to change the geometry or modify the models, and they most often act as black boxes. Here we use another category of commercially available software like FEMLAB [ref. 8]. FEMLAB is developed specifically for solving not only NS equations or CFD but also other Partial Differential Equations (PDEs) where the user in principle can define and simulate any system of coupled PDEs. Practical physics and engineering problems in the area of fluid mechanics and structural mechanics can be solved with the software. The problem solved in this paper is quasi-steady two-dimensional flow of the continuous oscillation of an equilateral triangular cylinder in the uniform incoming flow. One of the main advantages of FEMLAB is that the user can focus on the model (PDE coefficients on the domain and the domain boundary) and does not have to spend much time on solving and visualization.

FEMLAB is a toolbox written in MATLAB [ref. 9]. It solves system of coupled PDEs of upto 32 variables. The specified PDEs may be non-linear (e.g. NS equations) and time dependent and act on a one-, two- or three-dimensional geometry. The PDEs and boundary values can be represented by two forms. The coefficient form is as follows:

$$d_a \frac{\partial u}{\partial t} - \nabla \cdot (c \nabla u + \alpha u - \gamma) + \beta \nabla u + a u = f \quad \text{in } \Omega \quad (1a)$$

$$n \cdot (c \nabla u + \alpha u - \gamma) + q u = g - \lambda \quad \text{on } \partial \Omega \quad (1b)$$

$$h u = r \quad \text{on } \partial \Omega \quad (1c)$$

The first equation (1a) is satisfied inside domain Ω and second (1b) –so called “generalized Neumann boundary”, and third (1c) –so called “Dirichlet boundary” equations are both satisfied on the boundary

Simulation of flow past an oscillating triangular cylinder using Finite Element Method

of the domain $\partial\Omega$, n is the outward unit normal, and is calculated internally. λ is an unknown vector-valued function called the “Lagrange multiplier”. This multiplier is also calculated internally and will only be used in the case of mixed boundary conditions. The coefficients d_a , c , α , β , γ , a , f , g , q and r are scalars, vectors, matrices or tensors. Their components can be functions of the space, time and the solution u . For a stationary system in coefficient form $d_a = 0$. Often c is called the diffusion coefficient, α and β are convection coefficients, a is the absorption coefficient, and γ and f are source terms.

The second form of the PDEs and boundary conditions is the general form:

$$d_a \frac{\partial u}{\partial t} + \nabla \cdot \Gamma = F \quad \text{in } \Omega \quad (2a)$$

$$-n \cdot \Gamma = G + \lambda \quad \text{on } \partial\Omega \quad (2b)$$

$$R = 0 \quad \text{on } \partial\Omega \quad (2c)$$

The coefficients Γ and F can be functions of the space, time, the solution u and its gradient. The components of G and R can be functions of the space, time and the solution u .

Given the geometry data, an initial finite element mesh is automatically generated by triangulation of the domain. The mesh is used for discretization of the PDE problem and can be modified to improve accuracy. The geometry, PDEs and boundary conditions are defined by a set of fields similar to the structure in the language C. A graphical user interface is used to simplify the input of these data. For solving purposes, FEMLAB contains specific solvers (like static, dynamic, linear and non-linear solvers) for specific PDE problems.

The incompressible fluid mechanics in two- and three-dimensions, is modeled within the incompressible Navier-Stokes (NS) application mode. The time-dependent NS equations can be written as:

$$\rho \frac{\partial u}{\partial t} - \nabla \cdot \eta (\nabla u + (\nabla u)^T) + \rho(u \cdot \nabla)u + \nabla p = F \quad \text{in } \Omega \quad (3a)$$

$$\nabla \cdot u = 0 \quad \text{in } \Omega \quad (3b)$$

In terms of the FEMLAB formulation, the NS equations are expressed by using the PDE coefficients according to the following table. Notice that the general form is used in this application mode:

Coefficient	Expression
d_a	ρ
Γ	$\begin{bmatrix} -\eta(\nabla u + (\nabla u)^T) \\ 0 \end{bmatrix}$
F	$\begin{bmatrix} F - \rho(u \cdot \nabla)u - \nabla p \\ -\nabla \cdot u \end{bmatrix}$

Simulation procedures and results

The cylinder is initially positioned such that the frontal face is at -30 degrees angle to the undisturbed freestream. The body fit elements are created, using the built-in function inside FEMLAB, and refined such that there are always 12,000 elements in the computational field to insure accuracy. The example of the meshing (elements) for a particular wedge position is shown in figure 6.

The simulation takes place to get the streamline and the pressure acting on the lateral faces of the cylinder (both upper and lower). Then, from the pressure information, we can find the moment acting around the cylinder (Srirarom, 2003 [ref. 7]). It is the unbalance pressures (moments) on the upper and lower side faces of the cylinder that lead to the angular movement of the cylinder around the

geometric center. The example pressure contours are shown in figures 7 and 8, for the cylinder at +30 degree and -30 degree positions.

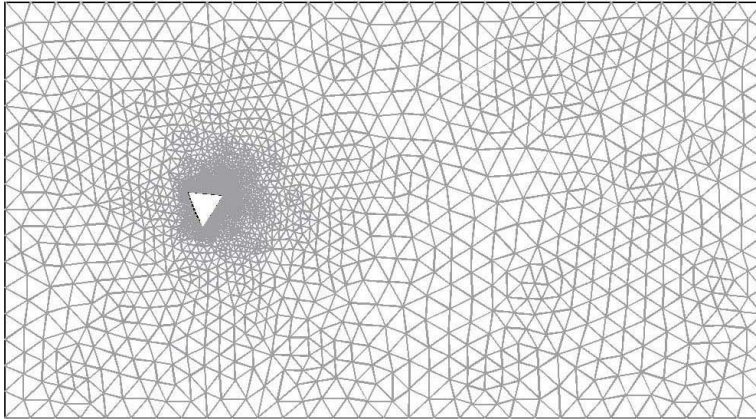


Fig. 6: Body-fitted element around the cylinder. The total number of element is always around 12,000.

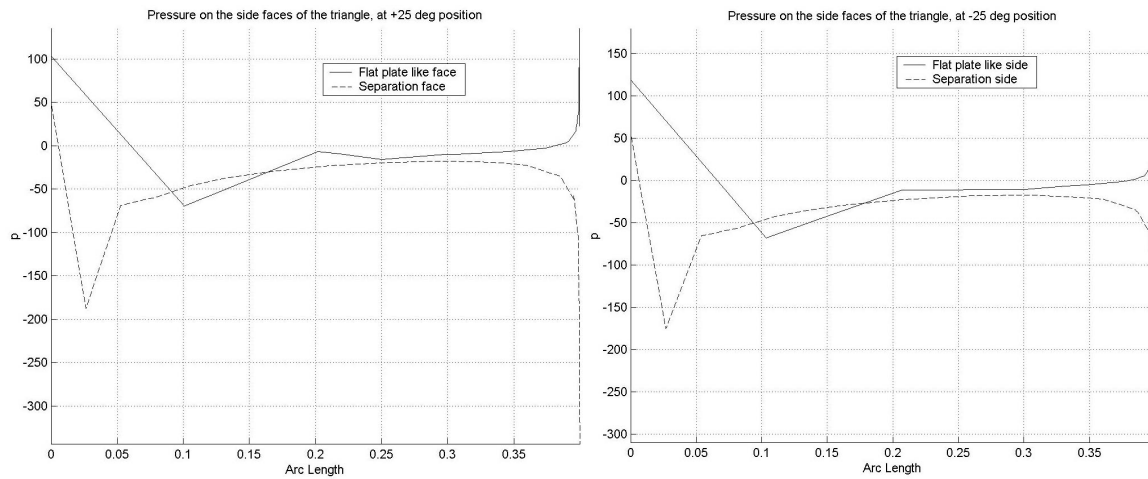


Fig. 7 (left) and 8 (right): Pressure acting on the lateral sides of the cylinder at +25 degrees (left) and -25 degrees (right) position. The flat plate like and separation faces are referred to their flow patterns. They are either upper or lower side faces according to cylinder position (see the simulation in Fig. 9).

From the pressure unbalance (and, hence, the turning moment), we can find the consequent angular displacement at the fixed time increment. Then, we redraw the geometry –rotate the cylinder to the new position, and do another computation for streamline and pressure distribution. The process is then repeated until the cylinder moves to another extreme position of +30 degree, then, the simulation stops. The termination of simulation at +30 degree is because, at this position, the flow pattern is the vertically flipped image of the initial position (-30 degree) already, and that, one can argue that the reverse process will take place. The enlarged streamline flow patterns from -30 degrees to +30 degrees positions are shown in figure 9 (a)-(m) below.

The FEMLAB result in figure 9 (a) shows that, on the windward side of the cylinder, a vortex is formed at the sharp edges of the cylinder during the initial phase (at lower face), whereas on the leeward side, the flow stays attached (at upper face). The size of the separated vortex (lower side) increases with the inlet velocity. Toward the rear of the cylinder on the leeward side (upper side), the sharp edge behind the cylinder creates another vortex –of the opposite sign to the windward one, and causes reverse flow. Behind the cylinder, the pair of windward and leeward vortices interact each other, are shed downstream by the main flow, disintegrated by viscosity and disappear.

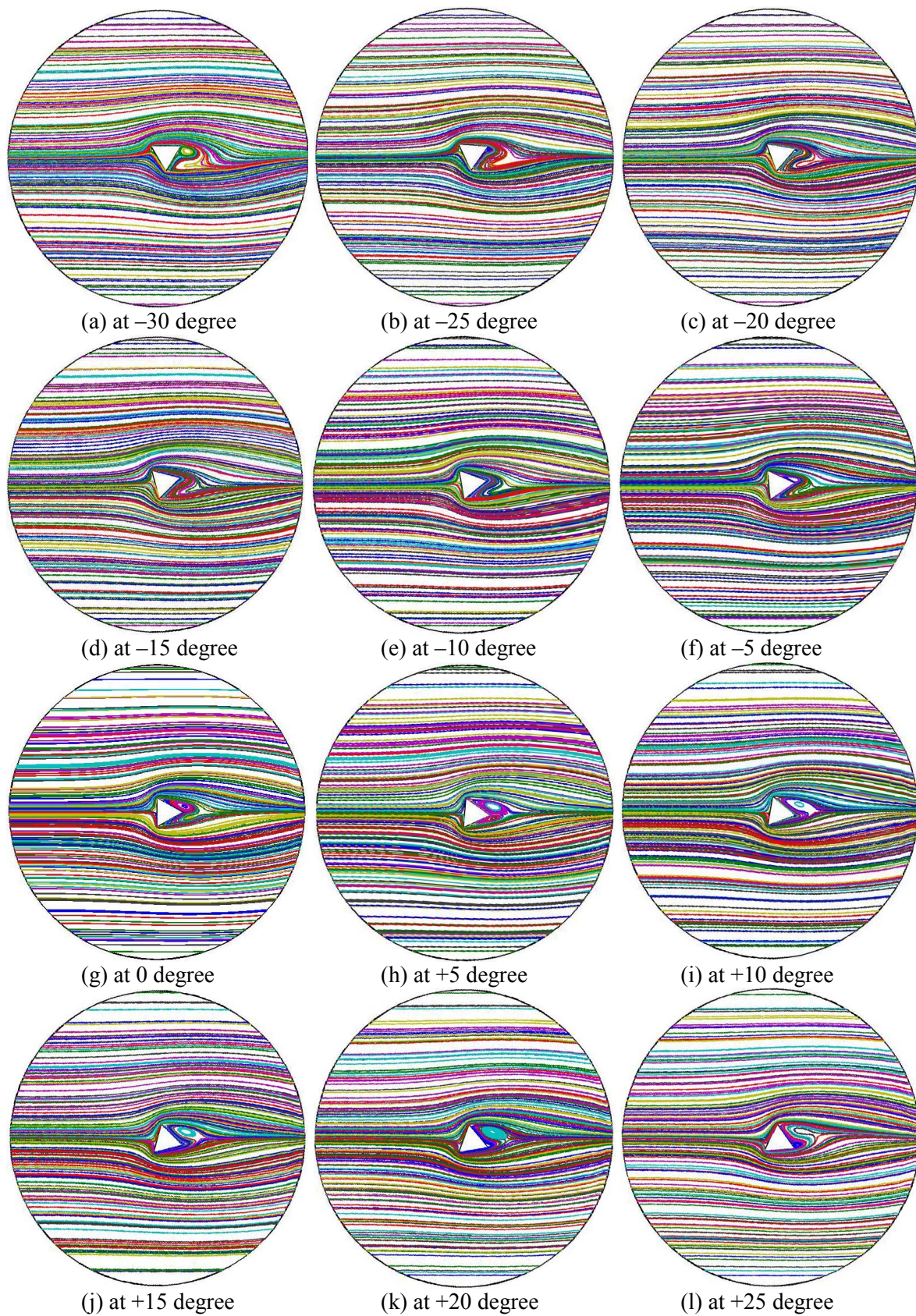
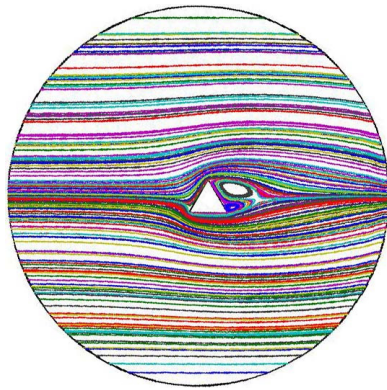


Fig. 9 (a)-(m): The simulation of the rotating equilateral triangular cylinder in clockwise direction. The Reynolds number based on cylinder's width = 1.



(m) at +30 degree

Fig. 9 (a)-(m) (Continued): The simulation of the rotating equilateral triangular cylinder in clockwise direction. The Reynolds number based on cylinder's width = 1.

As time goes on, as shown in figures 9(e), (f) and (g), the flow on the upper side becomes more diverging, and that the new separating vortex starts to form. The flow behavior on this upper side changes from flat plate like to separation like pattern. In the mean time, the flow on the lower side becomes more converging, and that the existing separating vortex is suppressed. The flow behavior on this lower side changes from separation like to flat plate like. The flow on both sides change their patterns, as the cylinder is at zero degree position, as shown in figure 9 (g), and even (h). Although, this series show only clockwise motion, the interchange of flow behavior also exists at zero degree position in the returning counterclockwise motion, as well.

Insofar the simulation results have been showing the self-excited oscillation behavior of the cylinder. It is found from the prior experiments that, the agreement extends not only the overall oscillating characteristics, but also, the flow details. In figures 10, the comparison of the flow patterns between the one from the real flow phenomena in water and another one from the present CFD simulation (FEMLAB result), at the same angular position of +25 degree. The shape of the stretched bubbles in the water shown in the left picture depicts the flow direction, and agree well the streamline in the simulated CFD result shown in the right picture. In figures 11, the comparison of the flow patterns between the one from the real flow phenomena in water and another one from the present CFD simulation, at the same angular position of +25 degree. The path of the stretched bubbles in the water shown in the left picture depicts the flow separation, at the lower tip, and agree well the streamline in the simulated CFD result shown in the right picture. As anticipated, figures 11 appear as the vertically-flipped image of figures 10.



Oscillating triangle at 25deg position, pressure contour and streamlines

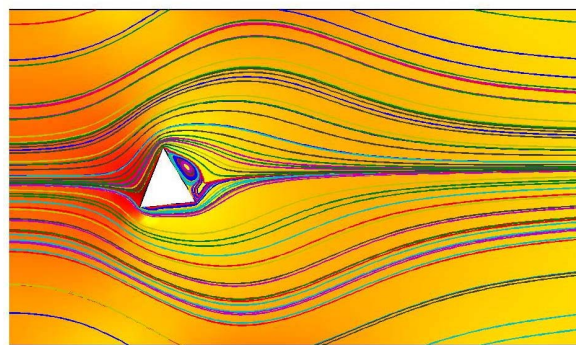


Fig. 10: Comparison of the flow pattern between the real flow phenomena in water and the present CFD simulation, at the same angular position of +25 degree..

Simulation of flow past an oscillating triangular cylinder using Finite Element Method

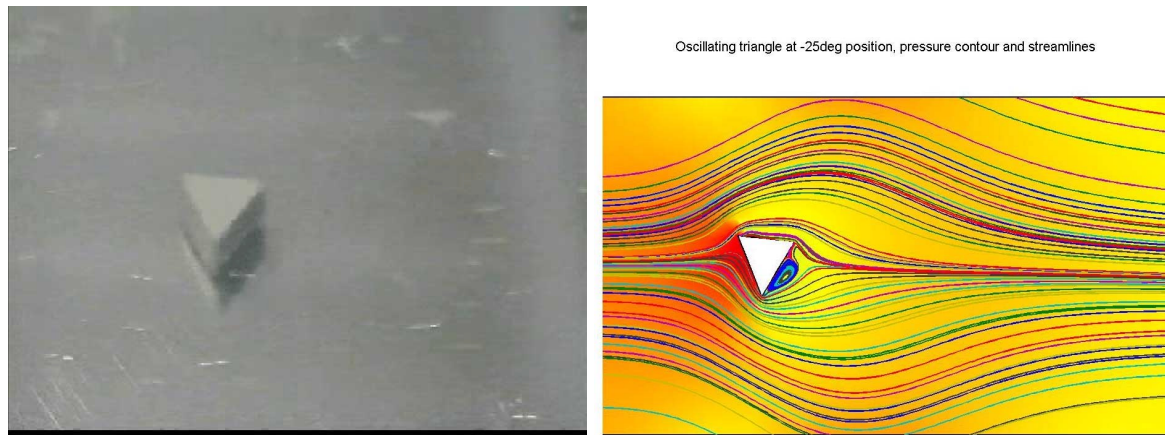


Fig. 11: Comparison of the flow pattern between the real flow phenomena in water and the present CFD simulation, at the same angular position of ± 25 degree.

Conclusions

The present CFD study of a simplified quasi-steady flow past an oscillating triangular cylinder found an interesting self-excited oscillation of the cylinder as observed in experimental investigation before. Under the simple uniform incoming flow, the interchange flow pattern, at certain flow velocity, was found. The results from Finite Element Method (FEMLAB software) appeared to agree with the proposed underlying physical explanation of the phenomena. It is the unbalance force acting on the cylinder's side faces that causes such movement. On one side, the flow will be flow-past-flat-plate like, whereas the other side will be flow-past-sharp-edge like. When the cylinder rotates, these mechanisms switch side interchangeably, and bring the cylinder to continuous oscillation. The qualitative agreement between the theory, simulation and the experiments are pronounced. This study of the flow past an oscillating triangular cylinder could be further enhanced by considering more robust fluid-solid interaction, incoming flow fluctuation and non-Newtonian effect.

References

1. Nagashima, T. & Hirose, T., "Potential Flow around Two Dimensional Isosceles Triangular Cylinder Subjected to Uniform Flow from Base Surface", Journal of Japan National Defense Agency.
2. Hu, C.C, Miao, J.J. and Chou, J.H., "Instantaneous Vortex-shedding Behaviour in Periodically Varying Flow", Proc. Roy. Soc. Lond. A, Vol. 458, pp. 911-932, 2002.
3. Sakamoto, H., Takai, K., Alam, M.M. and Moriya, M., "Suppression and Characteristics of Flow Induced Vibration of Rectangular Prisms with Various Width-to-height Ratios", in "Fluid Structure Interaction", (eds.) Chakrabarti, S.K & Brebbia, C.A., pp. 67-76. WIT Press, 2001.
4. Naudascher, E. & Wang, Y., "Flow-induced Vibrations of Prismatic Bodies and Grids of Prisms", J. Fluids and Struct., Vol. 7, pp. 341-373, 1993.
5. Nakamura, Y. & Nakashima, M., "Vortex Excitation of Prisms with Elongated Rectangular, H and $\perp$ Cross-sections", J. Fluid Mech., Vol. 163, pp.149-169. 1986.
6. Srigrarom, S., "Self-Excited Oscillation of Triangular Cylinder", Master Thesis, Department of Aeronautics and Astronautics, University of Washington, 1998.
7. Srigrarom, S., "Self-Excited Oscillation of Equilateral Triangular Wedge", Proceedings of the IUTAM Symposium on Fluid-Structure Interactions, New Brunswick, New Jersey, June 2-6, 2003, Vol.1, 2003.
8. COMSOL, AB, FEMLAB Version 2.2, Reference Manual, November 2001
9. The Mathworks, Inc., MATLAB Version 5.3, Manual, 1998.