International Journal of Scientific World, 1 (2) (2013) 37-43
©Science Publishing Corporation
www.sciencepubco.com/index.php/IJSW

Hydrodynamic analysis of flow around square cylinders and determination of the strouhal number as a function of angles of attack and reynolds numbers

Sabra Razughi , Parviz Ghadimi *, Saman Kermani
Department of Marine Technology, Amirkabir University of Technology, Tehran, Iran
**Corresponding author E-mail: pghadimi@aut.ac.ir*

Abstract

Hydrodynamic analysis of flow around square cylinders at different inclined angles of attach ranging from 0 to 45 at low Reynolds numbers, both in water and air, has been carried out using commercial software ANSYS CFX. Twenty different square cylinders were modeled at angles 0, 5, 10, 15, 20, 25, 30, 40 & 45 degrees. Results obtained by the hydrodynamic analyses were analyzed by Eureqa software to find a relation between non dimensional parameters. Strouhal number was determined to be a function of angle of attack and 1/Re in water and air. The obtained formulas were tested at two random angles of 33 and 50 against the results of Ansys-CFX. Comparison displayed good agreement which is demonstrative of the acceptability of the obtained relationships.

Keywords: *Square Cylinders; Strouhal Number; Reynolds Numbers; Vortex Shedding.*

1 Introduction

One of the most challenging problems in turbulent flows is vortex shedding phenomenon. When a uniform flow passes arond a Bluff Body, periodic vortices will shed above a critical Reynolds number, as illustrated in Fig.1. During this process, pressure fluctuations will develop on the cylinder surfaces which are dipole noise resources. Therefore, it is important to study these noise polluters [1, 2].

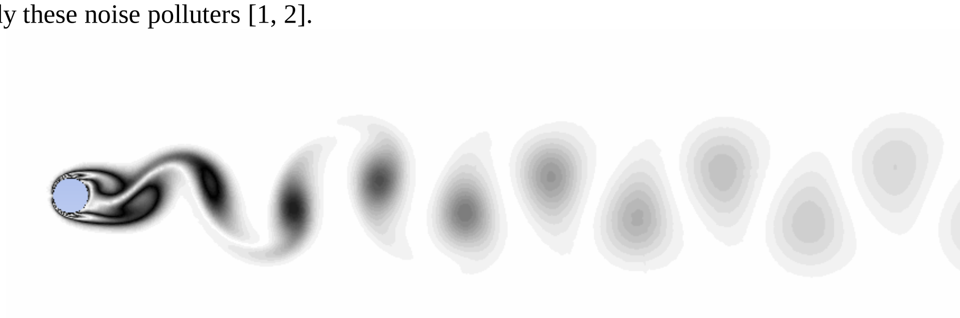

Fig.1: Vortex shedding phenomenon around a circular cylinder.

This phenomenon was first studied by Strouhal in 1878 [3]. He showed that the fundamental frequency of vortex shedding sound is equal to $f_0 = S_{t_0} U_0 / d$ in which fundamental Strouhal number is $S_t = .2$, U_0 is the free stream velocity and d is the cylinder's diameter. The fundamental frequency is the lowest frequency before resonance. Subsequently, Curle [4] in 1995 showed that these noises will intensify at low Mach numbers.
Many researchers have modeled these objects as cylinders with different cross-sections such as square, circular, and diamond, among which square and circular cylinders were the most interested sections. [5, 6, 7]
Williamson [8], Sohankar [9] and Lue [10] are the pioneers in this field of research. Cox [3] also in 1998 studied circular cylinder and showed that choosing the right turbulence model is highly important. Dalton and Zeng [11] in 2003

numerically studied the flow past square and diamond cylinders at Reynolds numbers of 200 and 1000. Dutta, Panirgrahi and Muralidhar [12] also in 2008 experimentally studied flow around square cylinders at four angles 0, 22.5, 30, and 45 degrees and for two aspect ratios of 16 and 28. King and Pfizenmayer [13] researched the sound generated by rigid cylinders in cross flow experimentally in 2009. Another experimental work has been carried out by Huang, Lin and Yen [14] in 2010. They studied flow regimes around a square cylinder between the angles of 0 to 45 degree and showed that the angle of 20 has a maximum Strouhal Number. Yoon et al [15] also in 2010 worked on flow around square cylinder at different angles of attack.
In this study, RANS' equations have been solved using commercial ANSYS CFX software Strouhal Numbers related to the square cylinders between angles 0 to 45 degrees have been studies in both air and water.

2 Governing equations

The commercial software ANSYS CFX uses RANS methods in order to solve the hydrodynamic problems. RANS method is the Navier-Stockes time-averaged equations. The continuity, momentum and energy equations in this method are as bellows:
The continuity equation:

$$\frac{\partial \overline{u_k}}{\partial x_k} = 0 \tag{1}$$

The momentum equation:

$$\frac{\partial}{\partial x_j}\left(\rho \overline{u_j}\,\overline{u_i}\right) = \frac{\partial}{\partial x_j}\left(-\bar{p}\delta_{ij} + \mu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) - \rho\overline{u_i' u_j'}\right) + \rho g_i \tag{2}$$

The energy equation:

$$\frac{\partial}{\partial x_j}\left(\rho \overline{u_j}\,\overline{He}\right) - \frac{\partial}{\partial x_j}\left(\frac{\lambda}{C_p}\frac{\partial \overline{He}}{\partial x_j} - \rho\overline{u_i' He'}\right) = 0 \tag{3}$$

As seen in these equations, Continuity equation is not changed, but additional terms appear in Momentum and Energy equations. These new terms are well known as REYNOLDS STRESSES and REYNOLD FFLUXES [16]:

$$\rho\overline{u_i' u_j'} = Reynolds\ stresses$$
$$\rho\overline{u_i' He'} = Reynolds\ fluxes$$

When a turbulent flow is to be solved, all Reynolds stresses and its transfer fluxes must be solved. In this manner, the number of unknown terms exceeds the number of equations and additional assumptions will be necessary. One of these assumptions is Eddy-Viscosity assumptions. Many turbulent flows use this assumption. In this model, Reynolds stresses are assumed to be similar to viscous stresses in laminar flows and it is in the form of (4) equation (4) [16].

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{4}$$

in which $\mu_t$is the eddy viscosity or turbulence viscosity and k is the kinematic turbulent energy and is obtained from below equation (5) [16].

$$k = \frac{1}{2}\overline{u_i' u_j'} \tag{5}$$

By introducing k into equation (4) for i=j, the resulting equation is also correct for incompressible flows.
By the same strategy, Eddy diffusivity hypothesis is applied for Reynolds fluxes.

$$-\rho\overline{u_i' He'} = \Gamma_{He_t}\frac{\partial \overline{He}}{\partial x_j} \tag{6}$$

Where Γ_{He_t} is enthalpy diffusivity and is related to eddy viscosity through equation (7).

$$\Gamma_{He_t} = \frac{\mu_t}{\sigma_{He_t}} \tag{7}$$

In the above equation, $\sigma_{He_t}$is the turbulence Prantle number for enthalpy? With these assumptions, momentum and energy equations in RANS method are as belows:

$$\frac{\partial}{\partial \mathrm{x}_j}\left(\rho \bar{u}_j \bar{u}_i\right) = \frac{\partial}{\partial \mathrm{x}_j}\left(-\overline{p_o}\delta_{ij} + \mu_{eff}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right)\right) + \rho g_i \tag{8}$$

$$\frac{\partial}{\partial x_j}\left(\rho \bar{u}_j \overline{He}\right) - \frac{\partial}{\partial x_j}\left(\underbrace{\left(\frac{\lambda}{C_p} + \frac{\mu_t}{\sigma_{He_t}}\right)}_{\Gamma_{He_{eff}}}\frac{\partial \overline{He}}{\partial x_j}\right) = 0 \tag{9}$$

in which μ_{eff} and $\Gamma_{He_{eff}}$are effective viscosity and diffusivity respectively and are obtained from their laminar and turbulent parts as in

$$\mu_{eff} = \mu_l + \mu_t \tag{10}$$

$$\Gamma_{He_{eff}} = \Gamma_{He_l} + \Gamma_{He_t} \tag{11}$$

Where Γ_{He} is obtained from equation (12):

$$\Gamma_{He} = \frac{\lambda}{C_P} \tag{12}$$

Due to modifying the pressure term in the buoyant flows, p_o is used instead of p which is given as

$$p_o = \mathrm{p} - \rho_{Bref} g_k x_k + \frac{2}{3}\rho k \tag{13}$$

It is clear that momentum and energy equations in turbulent flow with eddy viscosity hypothesis is the same as laminar flow equations, but μ_{eff} and $\Gamma_{He_{eff}}$are replaced by μ and Γ. For solving the above equations, only determining μ_t is necessary. Prantle in 1925 proposed equation (14) for determining this value [16].

$$\mu_t = \rho C_\mu l^2 \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) \tag{14}$$

Using different turbulent models, the problem will be solved completely.
In order to find the relations involved in hydrodynamic analysis, Eureqa software was applied. This software was developed by Lipson [17] working on self-contemplating robots that determines how to repair themselves, in Cornell University. The software searches through data and poses series of simple questions trying to explain the relationships between the data. The initial equations often are not the correct behavior and show to fail. Ultimately, the best equation will be selected and tested. This cycle repeats over and over [18].

3 Problem Set-up

3.1 Software validation

For validating the simulation, a square cylinder was choseen for modeling. The problem is a rigid square cylinder whose length is 3.39e-5 and is located within 10 length of cylinder from the inlet, top and bottom boundaries and 25 lengths far from the outlet. Laminar uniform flow crosses the cylinder with speed 68.4 m/s. Air at 25^0C is the working fluid, thus the corresponding Reynolds and Mach numbers are 150 and 0.2, respectively. Figure 2 shows the geometry of the problem.

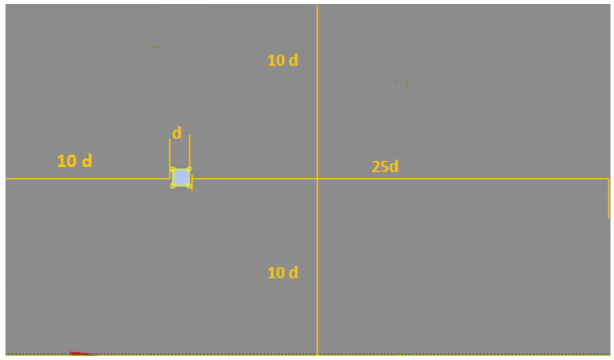

Fig.2: The geometry of problem.

The flow analysis was carried out using ANSYS CFX 13 as a hydrodynamic solver. Using CFX-Mesher module, the computational domain was divided into 227000 elements, in which each element size was 2e-6. Also finer mesh were used near the cylinder.

The initial condition was assumed to be Steady State, t=0 s and gauge pressure = 1atm. In order to increase the convergence speed and accuracy, the problem was solved in steady state manner, first. Then, the results were used as an initial condition for a transient run. By choosing 1e-7 [s] as a time step, the transient serial runs were started using 4 processor system. The results were extracted after the flow showed to be stationary (after 10 steady periodic oscillations).

Based on Sukri et al.'s study [19], the maximum C_L of cylinder is 0.2824. ANSYS CFX determines this value as 0.267. So, the percentage of error will be 5.4%.

$$\frac{0.2824 - 0.267}{0.2824} \times 100 = 5.4\% \qquad (15)$$

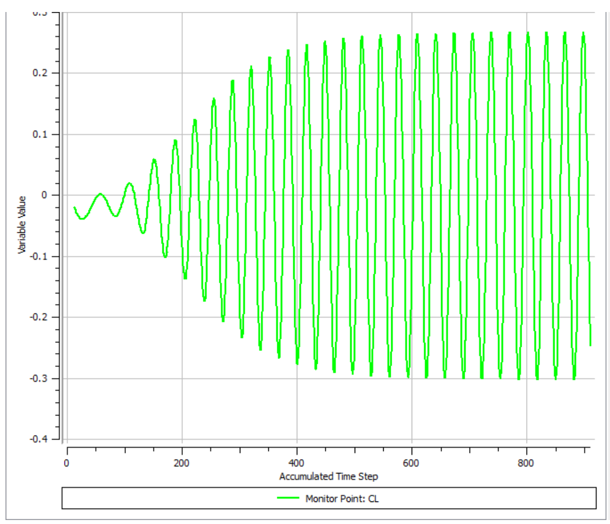

Fig.3: Lift coefficient of cylinder at Reynolds Number of 150 and Mach number of 0.2.

3.2 Problem definition

Similar domain was used to simulate 20 cylinders at angles 0, 5, 10, 15, 20, 25, 30, 35, 40 and 45 in air and water flows. First 10 cylinders have the length 3.39 e-5m and others have the length 1.34e-5 [m]. The coordinates were located in the western south corner of domain in which X axis is parallel to the flow and Y axis is perpendicular to it. Normal speed was used as an Inlet boundary condition having values 68.4 [m/s] for air and 10 [m/s] for water. The Outlet boundary condition was set to average static pressure of 0 atmospheres. Top and bottom boundaries were conditioned inlet with the parallel speed to X axis equal to the inlet speed value. No slip wall was applied on cylinder's surfaces. The steady state, gauge pressure of 0 atmosphere and t=0 [s] were considered as initial conditions.

4 Discussion of results

After ten periodic stable oscillations, CL results were considered. Figures 4 and 5 show the CLcurves for each cylinder at ten angles, respectively for water and air.
For determining that relationship between parameters in this hydrodynamic problem, parameters listed in table 1 were considered as involving factors.

Table 1: Involved parameters in the problem.

T	$\boldsymbol{\theta}$	L	V	$\boldsymbol{\mu}$	$\boldsymbol{\rho}$
Time[s]	degree	Cylinder Length[m]	Speed of flow[m/s]	Dynamic viscosity [Kg/ms]	Density[Kg/m3]
T	-	L	LT-1	ML-1T-1	ML-3

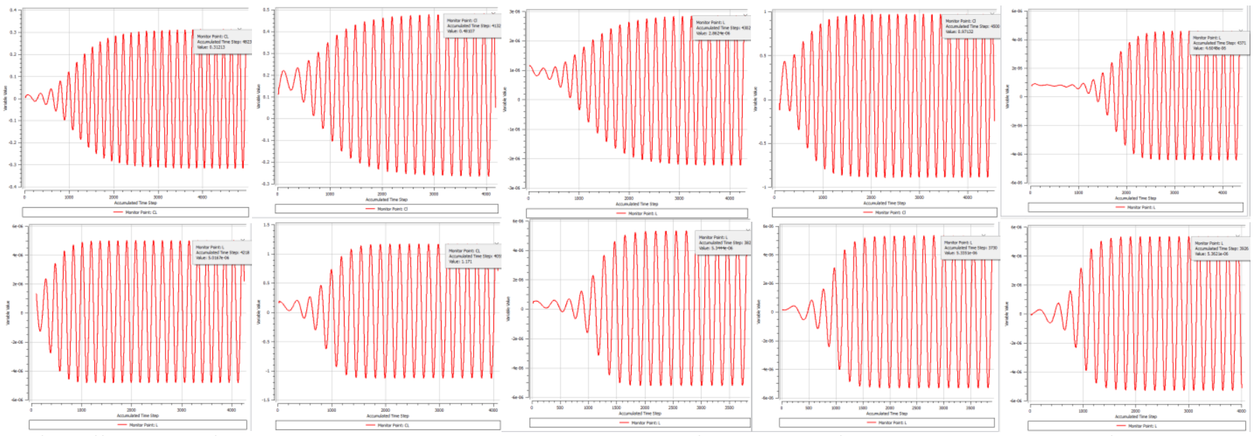

Fig.4: Lift Coefficient value for square cylinders surrounded by water at ten angles of attack ranging from 0 to 45 degree by a step of 5, respectively from left to right.

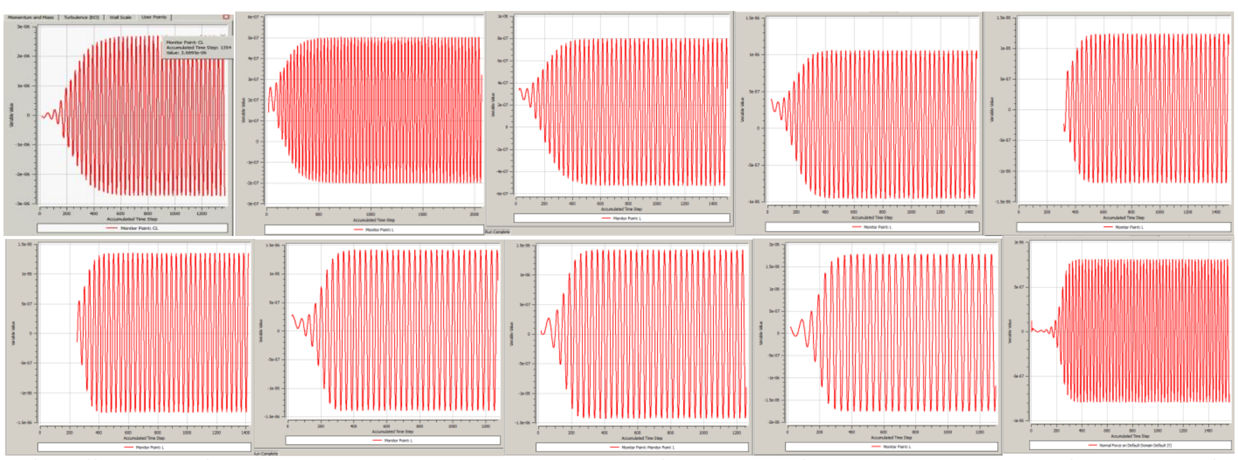

Fig.5: Lift Coefficient value for square cylinders surrounded by air at ten angles of attack ranging from 0 to 45 degree by a step of 5, respectively from left to right.

Using different relations from fluid mechanics [20], the independent π's can be found as follows:

$$\pi_1 = (T)^a.(L)^b.(\mu)^c.V \quad (15)$$

$$\pi_2 = (T)^a.(L)^b.(\mu)^c.\rho \quad (16)$$

So

$$\pi_1 = (T)^a.(L)^b.(ML^{-1}T^{-1})^c.LT^{-1} \quad (17)$$

$$\Rightarrow a = 1, b = 1, c = 0 \quad (18)$$

$$\Rightarrow \pi_1 = Strouhal\ Number \quad (19)$$

Therefore, it is concluded that

$$\pi_2 = (T)^a . (L)^b . (ML^{-1}T^{-1})^c . ML^{-3} \tag{20}$$

$$\Rightarrow a = 1, b = -1, c = 0 \tag{21}$$

$$\Rightarrow \pi_2 = \frac{1}{Reynolds\ Number} \tag{22}$$

$$\pi_3 = \theta \tag{23}$$

$$\therefore Strouhal = f\left(\theta, \frac{1}{Reynolds}\right), \qquad \square$$

Now, it is known that Strouhal is a function of Reynolds number and cylinder's angle. Table 2 shows the Strouhal number for each cylinder at different angles of attack and Reynolds number, in air and water.
Eureqa software has been applied to find the relation between Strouhal number, Reynolds and cylinders angle. Figure 5 shows the process of finding the formula in a glance.

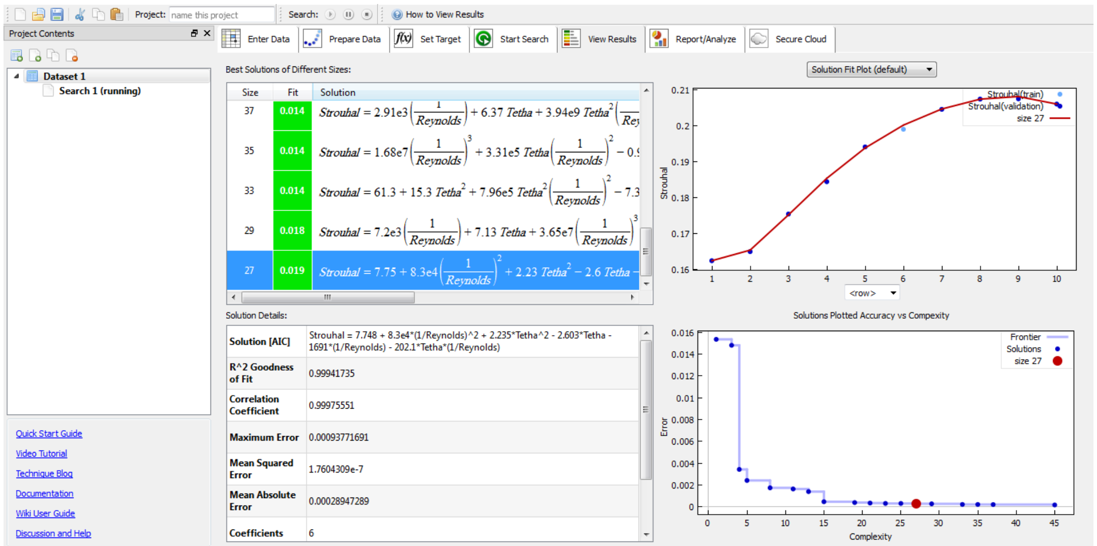

Fig.6: Relationship finding process by Eureqa Software.

Based on this software, relations (A) and (B) were extracted for Air and Water, respectively,

$$Strouhal = 7.748 + 8.3e4 \times \left(\frac{1}{Reynolds}\right)^2 + 2.235 \times \theta^2 - 2.603\theta - 1.69e3\left(\frac{1}{Reynolds}\right) - 202.1\theta\left(\frac{1}{Reynolds}\right) \tag{A}$$

$$Strouhal = 6.651 + 1.1e7\left(\frac{1}{Reynolds}\right)^3 + 929\theta^2\left(\frac{1}{Reynolds}\right) - 1465\theta\left(\frac{1}{Reynolds}\right) - 2.193e5\left(\frac{1}{Reynolds}\right)^2 \tag{B}$$

Table 2: Strouhal Number values for square cylinder at different angles of attack.

Air			Water		
θ	Reynolds	Strouhal	*θ*	Reynolds	Strouhal
0	150	0.154879	0	150	0.162424242
5	162.5026	0.167789	5	162.5026	0.164964726
10	173.7684	0.173984	10	173.7684	0.164964726
15	183.7117	0.189688	15	183.7117	0.175404627
20	192.2569	0.198511	20	192.2569	0.18439979
25	199.2569	0.205823	25	199.3389	0.19897699
30	204.9038	0.199124	30	204.9038	0.204522239
35	208.9093	0.203016	35	208.9093	0.207361796
40	211.3248	0.174559	40	211.3248	0.20745439
45	212.132	0.280362	45	212.132	0.2059328

In order to test the obtained formulas, two different cases at 33 and 50 degrees were modeled and Strouhal numbers were calculated both from the formula and also from the software result. Four cases were run in air and water for the purpose of checking the obtained formulas. Table 3 shows the Strouhal numbers which were calculated by the formula and obtained from the software.

Table 3: Strouhal Number values for square cylinder at different angles of attack, obtained from formula and CFX.

	Strouhal number in Air			Strouhal number in water		
	By formula	By CFX	error	By formula	By CFX	error
33	0.2074	0.2077	0.1%	0.213	0.2048	4%
50	0.2005	0.2053	2.3%	0.2097	0.2063	1.6%

Based on the results of table 3, the errors are below 5% and the obtained formula is correct both in air and water at each angle of attack.

5 Conclusion

ANSYS CFX 13 has been used to solve the hydrodynamic flow around square cylinders at angles of attach ranging from 0 to 45 degrees at low Reynolds numbers, both in air and water. Twenty different cases were modeled and the Strouhal numbers were extracted after 10 stable oscillations. By using some relations of fluid mechanics, Strouhal is found to be a function of cylinder's angle and Reynolds inverse. Therefore, by applying Eureqa software 2, relations were obtained for Strouhal number as a function of cylinder's angle and 1/Re. Subsequently, the results were checked for two angles of attach of 33 and 50 degrees both in air and water and it was proved that the results of the derived formulas are in good agreement with the computed results of the Ansys-CFX software at each of the considered angle.

References

[1] Doolan, C.J'."Computaional bluff body aerodynamic noise prediction using a statistical approach".J.AppliedAcoustics.September 2010.
[2] INASAWA, Ayumu; Nakano, Takuya; ASAI, Masahito."Development of wake voritices and the associated sound radiation in the flow past a rectangular cylinder of various aspect ratio".
[3] Cox, Jared et al."Computation of sound generated by viscous flow over a circular cylinder".NASA Technical Memorandum 110339. March 1997.
[4] Curle, N., The influence of solid boundaries on aerodynamic sound.Proc Roy SocLond; A231:505–14, 1955.
[5] Csiba, L.A., Martinuzzi.,J.R.,"Investigation of bluff body asymmetry on the properties of vortex shedding" , J. Wind Engineering, August 2007.
[6] Anagnostopoulos, P., " Numerical study of the flow past a cylinder excited transversely to the incident stream. Part2: timing of vortex shedding, Aperiodic phenomena and wake parameters", J. Fluids and Structure, March 2000.
[7] Bently,J.P., Nichols,A.R., " The mapping of vortex fields around single and dual bluff bodies", Butterworth-Heinemann Ltd, 1990.
[8] Williamson, C.H.K.,Vortex dynamics in the cylinder wake, Annual Review of Fluid Mechanics 28(1996) 477–539
[9] Sohankar, A., Norberg, L. C., Davidson, Simulation of three-dimensional flow around a square cylinder at moderate reynolds numbers, Physics of Fluids11 (2), 288–306, 1999.
[10] Luo, S. C., Chew Y. T. and Ng Y. T., "Characteristics of square cylinder wake transition flows", Phys. Fluids, 15 (9), pp. 2549-2559. 2003.
[11] Dalton, C., Wu, Z., " Numerical Solutions of a Viscous Uniform Approach Flow Past Square and Diamond Cylinders", 5th International Symposium on Fluid Structure Interaction, Aeroelasticity, and Flow Induced Vibration and Noise, 2003.
[12] Dutta, S., Panigrahi, P. K. and Muralidhar, K."Experimental investigation of flow past a square cylinder at an angle of incidence", ASCE Journal of Engineering Mechanics, 2008.
[13] King,W.,F., Pfizenmaier,E., "An experimental study of sound generated by flows around cylinders of different cross-section" .J. Sound And Vibration, July 2009.
[14] Huang, R. F., Lin, B.H., Yen, S.C., " Time-averaged topological flow patterns and their influence on vortex shedding of square cylinder in cross-flow at incidence" J. Fluids and Structures, February 2010.
[15] Dong-Hyeog, Y., Kyung-Soo,Y., Choon-Bum, C., " Flow Past a Square Cylinder with an Angle of incidence", J. Physics Of Fluids, 2010.
[16] Cook, M. J.," An Evaluation of Computational Fluid Dynamics for Modeling Buoyancy Driven Displacement Ventilation", Montfort University, PHD Thesis, 1998
[17] Dubcakova, R, " Eureqa- Software Review", Faculty of Safety Engineering, VSB, Technical University of Ostrava.
[18] Keim, B., Download Your Own Robot Scientist. Available online:
[19] http://www.wired.com/wiredscience/2009/12/download-robot-scientist/
[20] Ali, S.M., Doolan, C. J., Wheatly, V."The sound generated by a square cylinder with a splitter plate at low Reynolds number".J. Sound and Vibration, 2011.
[21] White, F., "Fluid Mechanics", McGraw-Hill, 2003.