_____

Original Paper

_____

Numerical Study of Important Factors for a Vortex Shedder using Automated Design Cycle

Su Myat Nyein[1], He Xu[2]

[1,2]College of Mechanical and Electrical Engineering, Harbin Engineering University
145, Nantong Street, Nangang District, Harbin, 150001, P. R. China, [1]sumyatnyein.me@gmail.com,
[2]railwaydragon@gmail.com

Abstract

The good performance of a vortex shedder is defined by strong and stable vortex generated under the condition of most linearity in Strouhal number and low power loss. In this paper, the flow past a bluff body of circular cylinder with a slit normal to the flow has been analyzed focusing on drag coefficient, linearity of Strouhal number and flow resistance (K-factor). The ANSYS/FLUENT package is used for flow simulation and the integration method of computational code to iSIGHT platform is employed for automated design cycle. This study results the design with (0.20~0.267) blockage ratio and 0.10 slit ratio as the best shedder for vortex flowmeter and this results are in well agreement with the experiment. As the combination of GAMBIT, FLUENT, and iSIGHT substitutes the design parameters automatically according to the input data, this method designs effectively the vortex shedder with less design cycle time and low manufacturing cost eliminating the human intervention bottleneck.

Keywords: drag coefficient, linearity of Strouhal number, K-factor, integration of GAMBIT, FLUENT, and iSIGHT

1. Introduction

It is well known that the bluff body called vortex shedder is a primary element of a vortex flowmeter and its geometry and blockage ratio strongly affect on the flowmeter performance. The good performance of a vortex shedder is defined by strong and stable vortex generated under the condition of most linearity in Strouhal number and low power loss. Linearity means the Strouhal number is constant or least deviation over the range of Reynold number. In the first design the circular cylinder was used as vortex shedder but it was found that the circular shape cannot give the fixed separation point in the range of Reynold number [1]. As the linearity of Strouhal number requires the location of the separation point to be fixed regardless of Reynold number, the sharp-edges shedder designs have been invented in order to remove the shifts resulting from the changes in boundary layer[2, 3]. The attempts of many researchers on vortex shedder by numerical and experimental analysis can be found in the primary element section of the review by [4]&[5].

A new bluff body, circular with slit, introduced by [6, 7] is also one of the breakthrough in the vortex shedder innovative world. Igarashi.T. [3] experimentally investigated on new two shedder designs (circular with slit, triangular-semi circular cylinder) and trapezoidal (that is the shedder design used in the industry). He studied on the flow resistance (K-factor) of vortex shedder and linearity of the Strouhal number. His work showed that these two shedders with a slit can reduced the pressure loss by 50% compared to trapezoidal one. In addition, the two shedders are superior in linearity, regularity, sensitivity and rangeability to the trapezoidal cylinder. Other researchers also carried out experiments for this vortex shedder design [8-10]. J.F. Olsen and S. Rajagopalan [8] made a particular attention to Strouhal number/Reynold number relation and drag coefficient. This work results that the inclusion of a slit normal to the flow increase the strength of vortices shed and drag coefficients were significantly increased for the cylinder with a slit normal to the flow.

The vortex shedder of vortex flowmeter has been designed numerically and experimentally. For numerical analysis, it is well known that the best description of von Karman Vortex Street is made by Navier-Stokes equations. Thus, ANSYS/FLUENT package [11] based on numerical solution of Navier-Stokes equations has been adopted for simulation of the phenomena appearing in vortex flowmeter. However, it has never found to optimize the design of vortex shedder using the integration of computational flow analysis with iSIGHT .The present work has analyzed numerically on the flow past a circular cylinder with a slit normal to the flow focusing on drag coefficient, Strouhal number and flow resistance (K-factor). In the present investigation, a state-of-art CFD code, FLUENT is used for flow simulation and the integration of computational code to iSIGHT platform is employed for an automated design. iSIGHT [12, 13] is one of the flexible design tools released in the last few years and it can solve the problem for thousands of iterations. The combination of GAMBIT, FLUENT and iSIGHT substitutes the data automatically according to the input parameters. Therefore, it can be found that this integration method is an effective one to design the vortex shedder for vortex flowmeter with less design cycle time

and low manufacturing cost eliminating the human intervention bottleneck.

2. Computational Flow Analysis

2.1 Mean Flow Equation

The continuity equation and unsteady momentum equation are:

$$\rho \frac{\partial}{\partial x_i}\left(U_i\right) = 0 \tag{1}$$

$$\rho \frac{\partial}{\partial t}\left(U_i\right) + \rho \frac{\partial}{\partial x_j}\left(U_i U_j\right) = -\frac{\partial(p)}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu_{eff}\frac{\partial(U_i)}{\partial x_j}\right) \tag{2}$$

Where ρ is mass density (kgm-3), t is time(s), U_i and U_j are longitudinal velocities in i[th] direction and j[th] direction respectively.

2.2 Turbulent Model

The RNG $k-\varepsilon$ model is one of the $k-\varepsilon$ variants of RANS derived using a rigorous statistical technique (called renormalization group theory). It is similar in form to the standard $k-\varepsilon$ model, but includes the features that make this model more accurate and reliable for a wider class of flows than the standard $k-\varepsilon$ model.

The RNG $k-\varepsilon$ turbulence model is used in this case. The unsteady transport equations for k and ε can be written as:

$$\frac{\partial}{\partial t}\left(\rho k\right) + \frac{\partial}{\partial x_i}\left(\rho U_i k\right) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right) + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{3}$$

$$\frac{\partial}{\partial t}\left(\rho\varepsilon\right) + \frac{\partial}{\partial x_i}\left(\rho U_i \varepsilon\right) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_{eff}\frac{\partial\varepsilon}{\partial x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_k + C_{3\varepsilon}G_b\right) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \tag{4}$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε respectively. S_k and S_ε are user-defined source terms.

2.3 Time derivatives and Discretization Scheme

The pressure-velocity coupling is handled with the SIMPLEC pressure correction procedure. The scheme is fully implicit in time and of second order and enough small time steps are used to minimize the discretization error.

2.4 Boundary Conditions

Velocity at the inlet and pressure at the pipe outlet are specified as the boundary conditions for the present case. The velocity inlet profile is set to a constant value and the hydraulic diameter (D_h) and turbulent intensity (I) are used as turbulent quantities. The turbulent intensity (I) is calculated from the following formula.

$$I = 0.16(\text{Re}_{D_h})^{-1/8} \tag{5}$$

No-slip boundary conditions are applied to the surface of bluff body as well as to the top and the bottom of the domain.

2.5 Computational Domain and Grid Generation

The following figure shows the vortex shedder designed in this project. In order to avoid the unnecessary effects of the entrance and the exit, the bluff body is placed at 40D after the inlet and 25D before the outlet where D represents the pipe diameter. The computational mesh employed are non-uniform grids and obtained using the Quad/Tri-Pave meshing scheme and Quad-Map meshing Scheme in GAMBIT. The total number of grid cells is over 230,000 but there may slightly vary with different bluff body size.
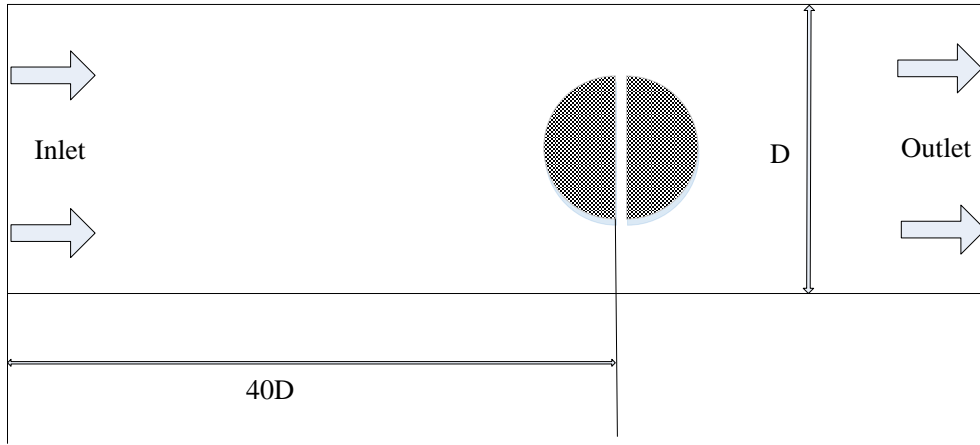
Fig. 1 Computational Domain

3. Validation of the Code

The flow past a circular with slit has been studied for the Reynold number of the pipe, ranged from $1.9x10^4$ to $2.5x10^5$ by using FLUENT. In the unsteady flow problems, meshing plays a significant role in the results accuracy. Due to this, an independence test of grid for a typical case of d=30mm and s/d=0.1at $Re_D = 1.9x10^4$ is conducted in the present study. Table 1 shows the meshing independence test for this study. It has found that Mesh III produce the results which are in well agreement with the experiment and no significant changes for further refinement Mesh IV. Therefore, the results are presented based on Mesh III. To validate the CFD code, the present simulation results are compared with the experimental data in Table 2.

Table 1 Meshing independence Test at $Re_D = 1.9x10^4 \sim 2.5x10^5$

Mesh	No. of Cells	C_d	K	St
I	133168	0.6939	0.224	0.142
II	149578	1.0609 (52.9%)	0.3232 (44.3%)	0.2525 (112.3%)
III	232672	1.5729 (48.3%)	0.41988 (29.9%)	0.2683 (6.3%)
IV	282772	1.5747 (0.11%)	0.4093 (2.52%)	0.2683 (0%)

Table 2 Validation of present results with existing experimental data (at $U_i=2m/s^2$)

Source	No. of Cells	C_d	K	St
Igarashi (experiment)[3]	-	-	0.48	0.26
Present Numerical	232672	1. 5729	0.41988(12.5%)	0.2683(3%)

Here, the coefficient of drag for bluff body is evaluated using the following relationship:

Drag coefficient,

$$C_d = \frac{Drag\ force}{\frac{1}{2}\rho U_i^2 A_p} \qquad (6)$$

Where, U_i^2 is velocity at the inlet and A_p is projected area of bluff body (m^2).

For pressure loss, the differential static pressure has been obtained at 415 mm upstream and 650 mm downstream from the vortex shedder because the pressure distributions at that region are nearly constant. The resistance coefficients, K-factor, for vortex shedder is

described by as follows:

$$K = (P_1 - P_2) / 0.5\rho U_i^2 \tag{7}$$

In this equation, P_1 and P_2 are static pressure at the upstream and downstream control points respectively. The Strouhal number of a vortex flowmeter is expressed as:

$$St = \frac{f\,d}{U_i} \tag{8}$$

Where f is the frequency of vortex shedding, d is the diameter of the bluff body.

4. Automated Design Methodologies

4.1 Vortex Shedder and Design Variables

Figure 2. shows the interested vortex shedder design and design variables for this study. In this paper, two design parameters are specified for design variables as follows: bluff body diameter (d), and slit to diameter ratio (s/d). The value ranges of the design variables (d) and (s/d) are $0.3 \le d \le 0.5$ and $0.1 \le s/d \le 0.3$ respectively.
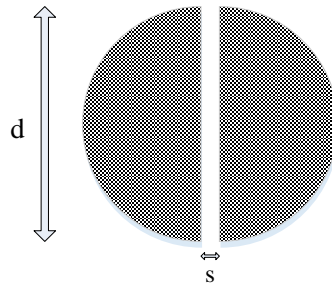
Fig. 2 Interested Vortex Shedder Design and design variables

4.2 Design Objective Functions

To be a good performance vortex shedder, it is essential to generate strong and stable vortex under the condition of most linearity in Strouhal number and low power loss. Therefore, three main points are made an attention for designing the vortex shedder: drag coefficient, flow resistance (K-factor) and linearity of Strouhal number. Here, the linearity of the Strouhal number is evaluated by means of error percentage. The error e is defined by the following equation:

$$e = \frac{1}{N}\sum_{i=1}^{N} \frac{\left|St_i - \overline{St}\right|}{\overline{St}} \tag{9}$$

4.3 Integration with iSIGHT for Automated Design

After validation of the CFD code, the automotive-design-cycle is used to design the vortex shedder. Figure 3. shows the iSIGHT automated design-evaluate-redesign cycle. The text file for designed geometry introduces into GAMBIT, create the model, and generate the mesh file. Then, FLUENT reads the mesh file generated from GAMBIT, get the solutions through the flow analysis setting and outputs the result file. EXCEL is for calculation the designed data using specific equations described above. Here, it is mentioned that the Strouhal number is calculated based on lift coefficient history file.
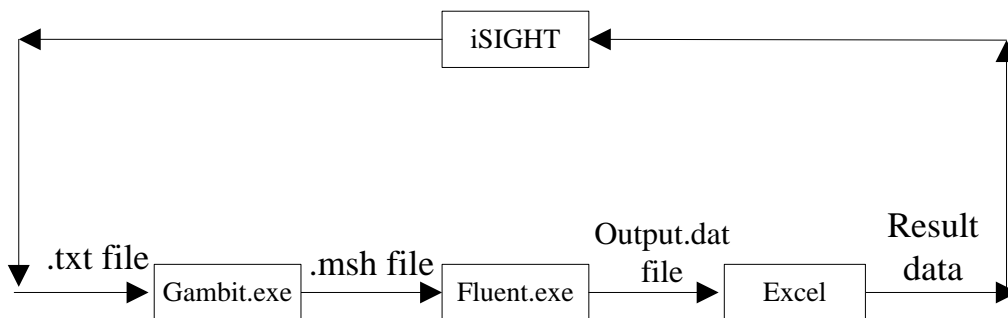

Fig. 3 iSIGHT Automated Design-Evaluate-Redesign Cycle

In this article, the FULL FRACTORIAL DOE technique is used to design. This technique specifies the number of levels for each factor and all combinations of all factors at all levels are studied. All possible factor interactions can be evaluated. The overall iSIGHT analysis process flow is outlined in Fig. 4., which shows the steps required to analyze the computational flow and calculate the outputs. The number of this step is the same the number of inlet velocity analyzed as GoFluent1 work for inlet velocity U1, GoFluent2 is for U2, and so on. The component 'Calculator' calculates the error percentage and then the process is repeated for the next input parameters. By this way, the process is executed continuously.
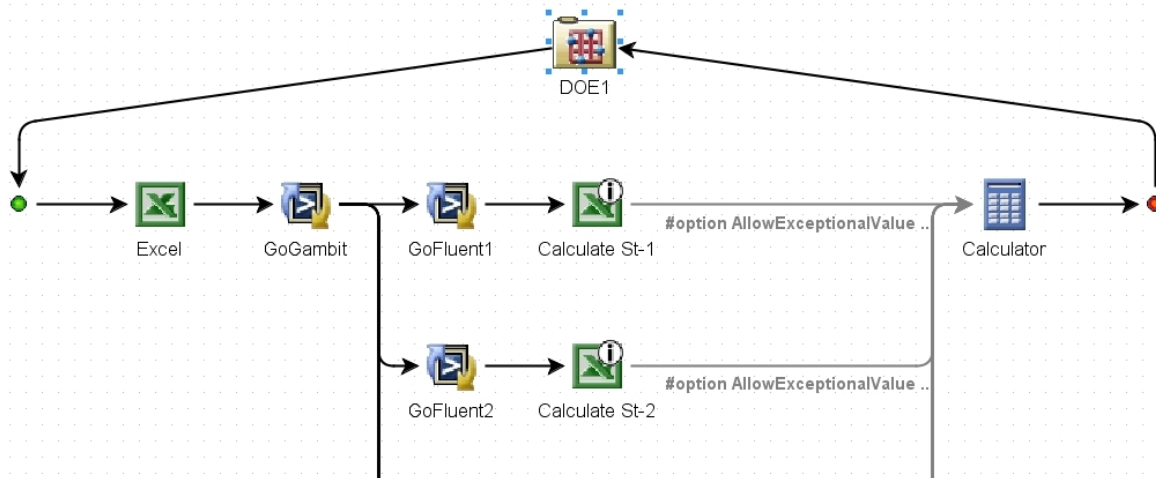
Fig. 4 iSIGHT design process starting at the green dot and finishing at the red dot

5. Automated Design Results

The essential factors of a vortex shedder has been analyzed numerically by using the integration method of computational fluid dynamic code, FLUENT to iSIGHT as Automated Design-Evaluate-Redesign Cycle. The geometry used in the present study is the shape of circular with slit and the designed parameters are shedder diameter (d) and the ratio of slit to diameter (s/d).The simulation results have been confirmed with experimental results and it has been found that the simulation results are well agreement with the experimental data. In the flow analysis, sufficient small time steps (60 time steps per one shedding cycle) are used in order to minimize the discretization error. Figure 5. shows the DOE results history generated by iSIGHT.

Fig. 5 DOE results history generated by iSIGHT

The linearity of Strouhal number against Reynold number was analyzed by changing the slit ratio (s/d) and bluff body diameter (d). According to the DOE automated results, it has found that s/d=0.10,0.20 are the ones generate the less variation in Strouhal number among the tested ratios for d=30mm while others give the higher error. For the shedder with 40mm diameter, s/d=0.10, 0.15 0.30 are scattered with less deviation   in the range of wide Reynold number and s/d=0.15,0.20.0.25 are for d=50mm. Figure 6. to 8. show the Strouhal number for each bluff body diameter at various slit ratio in the wide range of Reynold number. Overall, it can be seen that d=30 with s/d=0.1, d=40 with s/d=0.1 and d=50 with s/d=0.25 are the most linearity in Strouhal number with the error percentage of 1.3967%, 1.5054% and 0.9554% respectively. Figure 9. shows the variation in error (%) at different slit ratios for various vortex shedder diameters.
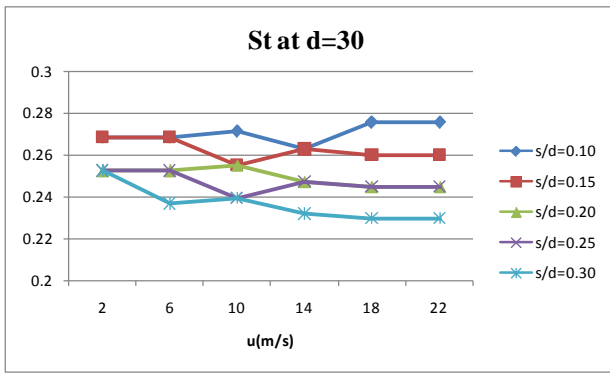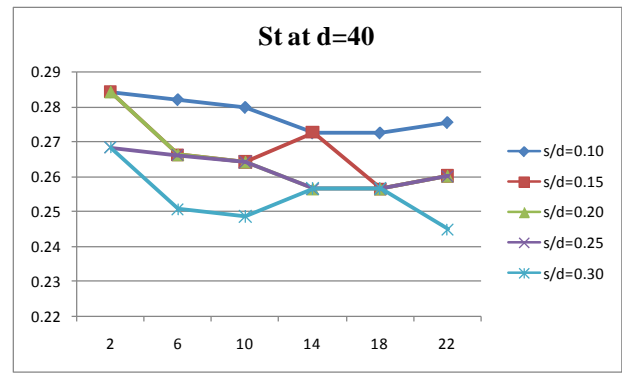
Fig. 6 St at various slit ratio (d=30mm)

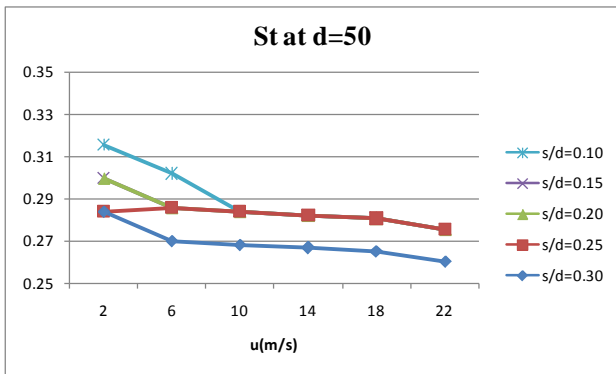**Fig. 7** St at various slit ratio (d=40mm)

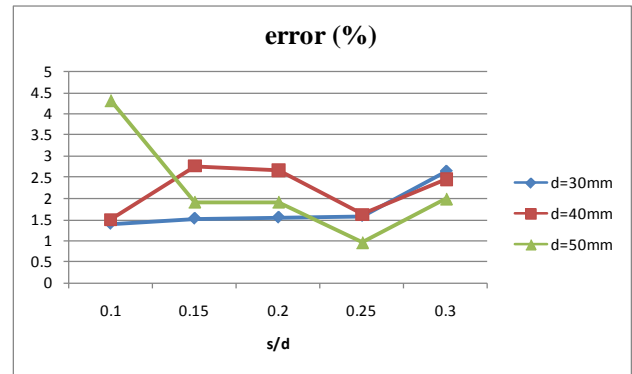**Fig. 8** St at various slit ratio (d=50mm)

Fig. 9 Variation in error (%) at different slit ratio (s/d) for various shedder diameters (d)

It is obvious that drag coefficient increases with an increase in shedder size as it is a function of projected area and velocity (see Eq. (6)). On the other hand, the big shedder size gives the large drag coefficient value as it has the large surface area. Although the C_d depends on flow rate, for the shedder of 40mm diameter with 0.1 slit ratio, Cd is nearly constant for the range of Reynold number from 1.9×10^4 to 2.5×10^5. It means drag coefficient is less dependent on mean velocity for the bluff body with 40mm diameter. In Fig.10. it can be seen clearly, the larger the shedder size, the increase the drag coefficient and also Cd increases with an increase in (s/d) ratio. Figure. 11 represents the relationship between Cd and slit ratio. This figure point the fact that the large slit gap cause the higher drag coefficient for the test range of Reynold number expect the highest one.
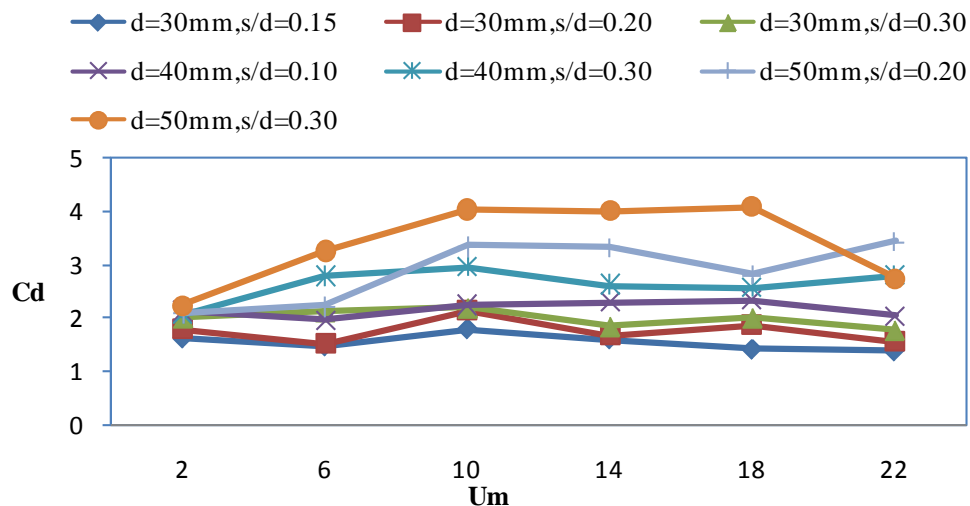

Fig. 10 Drag coefficient in the range of Reynold number for different bluff body size and slit
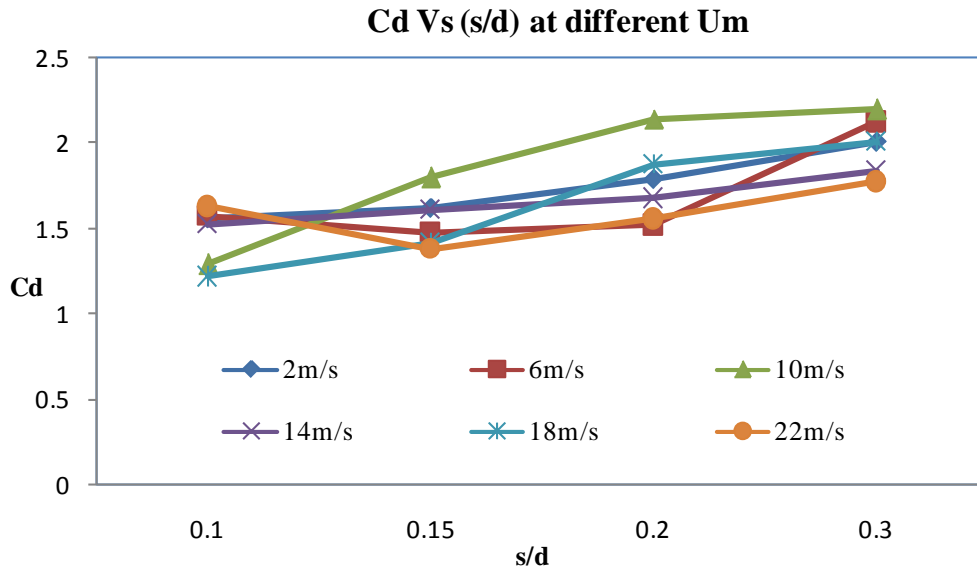
129

Fig. 11 The relationship between drag coefficient and slit ratio at different flow rate

For flow coefficient (K-factor) analysis, the results state that the resistance coefficient decreases with an increase in Reynold number for $Re_D \geq 10^5$ For 50mm diameter, however, this coefficient value ascends in highest velocity. And, the flow resistance of the shedder having 40mm diameter and 0.1 slit ratio is nearly constant regardless of Reynold number. The flow resistance values of the vortex shedder with various slit ratio and shedder diameter is plotted in Figure12. By the study of drag coefficient and flow resistance, it has been noticed that the design give the constant value of C_d also generates the linearity of K-factor. Thus, the present analysis support the relationship of K factor and C_d expressed by [3].
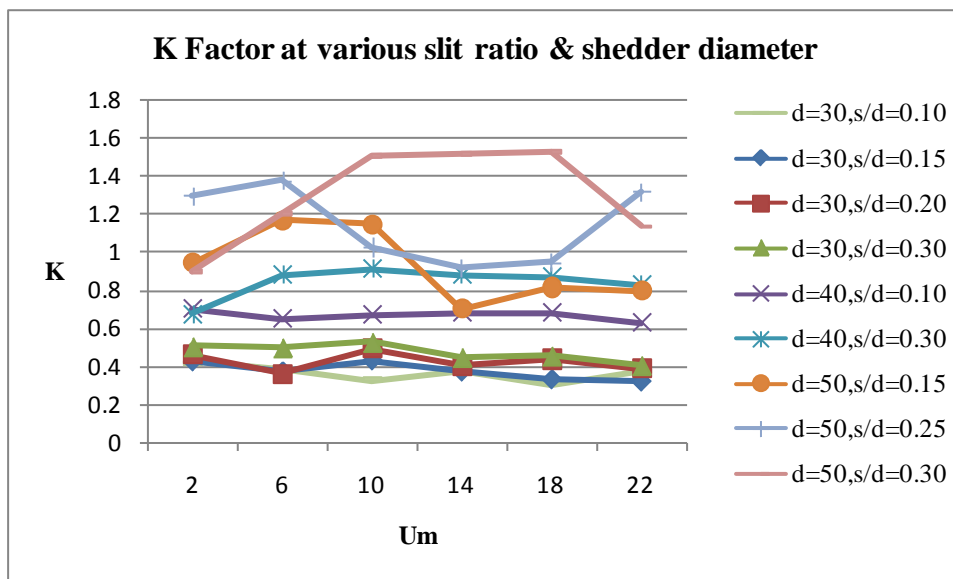

Fig. 12 Flow resistance at various slit ratio and shedder diameter

The performance of a vortex shedder is assessed by the generation of strong and stable vortex under the condition of most linearity in Strouhal number with low power loss. For the linearity, of 50mm diameter is the best shedder design. However, d=50mm with s/d=0.25 generate high value in K-factor. It means this parameter design generate high pressure loss. Thus, d/D=0.2~0.267 (i.e. d=30~40 mm) and s/d=0.1 is chosen as the most effective shedder design for vortex flowmeter.

6. Conclusion

The circular cylinder with slit of vortex shedder has been designed by varying the shedder diameter (30~50) mm and slit ratio (0.1~0.30) respectively. This numerical investigation is carried out with Automated Design-Evaluate-Redesign Cycle using iSIGHT. According to the results generated by iSIGHT, it can be concluded that

(1)   The RNG $k - \varepsilon$ turbulent model can generate the results as accurate as the experimental results with deviation of 3% in Strouhal number.

(2)   d/D =0.20~0.267 with slit ratio=0.1 is the most effective shedder for the vortex flowmeter.

(3)   At d=40mm, s/d=0.10, flow resistance is nearly constant regardless of Re_D.

(4)   For 30mm diameter, drag coefficient increases with an increase in (s/d) ratio for $Re_D \geq 10^5$ .But, for 40mm with 0.10 slit, C_d is nearly constant in the whole range of Reynold number.

(5)   These findings of present numerical analysis are in well agreement with the experiment.

(6)   Therefore, it can be found that this integration method is an effective one to design the vortex shedder for vortex flowmeter with less design cycle time and low manufacturing cost eliminating the human intervention bottleneck.

Acknowledgements

References

[1] E. Achenbach, 1968,"Distribution of local pressure and skin friction around a circular cylinder in cross-flow up to Re = 5X106", Journal of Fluid Mechanics, 34 (part 4), 625-639.
[2] Popiel C.O., Robinson D.I. and Turner J.T., 1993,"Vortex Shedding From a Circular Cylinder With a Slit and Concave Rear Surface ", Applied Scientific Research, F.T.M. Nieuwstadt (ed.), Advances in Turbulence IV, 51, 209-215.
[3] Igarashi T., 1999,"Flow resistance and strouhal number of a vortex shedder in a circular pipe", JSME International Journal, 42, 4, 586-595.
[4] Pankanin G. L., 2005,"The vortex flowmeter: various methods of investigating phenomena", MEASUREMENT SCIENCE AND TECHNOLOGY, 16, R1-R16.
[5] Venugopal A., Agrawal A. and Prabhu S.V., 2011,"Review on vortex flowmeter—Designer perspective", Sensors and Actuators A: Physical, 170, 8-23.
[6] Igarashi T., 1978,"Flow Characteristics around a Circular Cylinder with a Slit (1st Report, Flow Control and Flow Patterns)", Bulletinof the JSME, , 21, 154,
[7] Igarashi. T, 1982,"Flow Characteristics around a Circular Cylinder with a Slit (2nd Report, Effect of Boundarzy Layer Suction)", Bulletin of the JSME, , 25, No.207,
[8] Olsen J.F. and Rajagopalan S., 2000,"Vortex shedding behind modified circular cylinders", Journal of Wind Engineering and Industrial Aerodynamics, 86, 55-63.
[9] Wang Junwei, Yang Jinwen, He Yi and Bao Feng, 2012,"Experimental Study of Slit Cylinder Vortex Shedding in Circulating Water Channel", International Conference on Measurement, Information and Control (MIC)
[10] Peng B.H. , Miau J.J., Bao F., Weng L.D., Chao C.C. and Hsu C.C., 2012,"Performance of vortex shedding from a circular cylinder with a slit normal to the stream", Flow Measurement and Instrumentation 25, 54-62.
[11] 2006,"FLUENT 6.3 User's Guide",
[12] 2003,"iSIGHT User's Guide", version 8.0,
[13] 2009,"iSIGHT 4.0 Getting Started Guide",