Original Paper

Parametric Optimization of Vortex Shedder based on Combination of Gambit, Fluent and iSIGHT

Su Myat Nyein[1], He Xu[2], Hongpeng YU[3]

[1,2,3] 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, [3]yhp_1990@163.com

Abstract

In this paper, a new framework that works the automatic execution with less design cycle time and human intervention bottlenecks is introduced to optimize the vortex shedder design by numerical integration method. This framework is based on iSIGHT combined with the pre-processor GAMBIT, and flow analysis software FLUENT. Two vortex shedders, circular with slit and triangular- semi circular cylinder, are employed as the designed models to be optimized, and DOE driver is used for optimization. According to the essential properties of a vortex shedder, it has found that the best diameters are 30mm for circular cylinder with slit and 30 to 35 mm for tri-semi cylinder. For slit ratio, 0.1 and 0.15 are the optimized values for circular with slit and tri-semi cylinder respectively. And it is found that these optimal results generated by DOE automated design cycle are in well agreement with the experiment.

Keywords: Optimization, vortex shedder, how to integrate the GAMBIT and FLUENT to iSIGHT

1. Introduction

Many researchers have been analyzed on the flow past various shedders to achieve the best shedder designs [1-9]and some have been worked to find out the optimal dimension for a specific design [10-15]. By the past researches, it can be seen that most are experimental analyses and some have been observed by numerical method. It has never found, however, that the combination of the CFD performance analysis software and the optimization design platform is employed for numerical study. Obviously, it is too much cost for experimental analysis and the numerical analysis makes time waste for repeating the set of instruction again and again. Thus, it is needed to find out the new framework that works the automatic execution with less design cycle time and low manufacturing cost eliminating the man-made-errors.

Today's engineers use many varieties of design and simulation tools. Often parameters and results from one software package are used as inputs for the next software. Manual inputting the data can lead to reduced efficiency, slow product development, and human error. ISIGHT [16, 17], a generic software shell, can solve these issues. It provides a suite of visual and flexible tools to set up and manage computer software (including commercial CAD/CAE software) required to execute simulation based design process, internally developed programs, and excel worksheet. ISIGHT uses advanced techniques like Optimization, Design of Six Sigma Approximations and Design of Experiment (DOE). The relationships between parameters and results are easily understood and assessed, leading to the best possible design decisions. It enables to reduce design cycle time and manufacturing cost, and significantly improves product performance, quality, and reliability.

This paper devotes on how to integrate the pre-processor GAMBIT and the fluid dynamic code FLUENT to iSIGHT platform for automated design-evaluate-redesign-cycle to achieve the design optimization. This automated cycle involves the iterative processing of input files, the running of one or more simulation programs, and the analysis of output files. The cycle continues until the design deadline is reached and the best design that satisfies the requirements and meets design constraints is chosen by the design engineer. Although the iSIGHT platform has been used for many kinds of optimization such as aerodynamic optimization [18-21], it has been never found in vortex flowmeter design optimization. Therefore, it is very useful for practical engineering application and can be an efficient method to develop the new designs for vortex shedder.

2. Optimization Methodologies

2.1 Optimized Shedder Design and Variables

The interested vortex shedder designs and design variables ranges for optimization are presented in Table 1. In this paper, two design parameters, bluff body diameter (d), and slit to diameter ratio (s/d), are specified to optimize. The computational mesh are

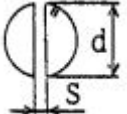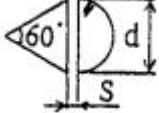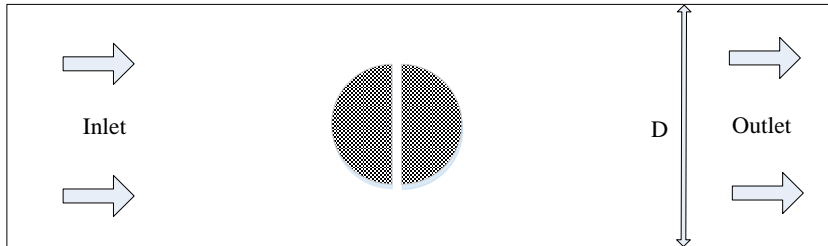obtained by using the Quad/Tri-Pave meshing scheme and Quad-Map meshing Scheme in pre-processor GAMBIT.

Table 1. Interested Vortex Shedder Design and Design Variables

Vortex Shedder	d (mm)	s/d
	30~ 50	0.1~ 0.3
	30~ 50	0.1~ 0.3

$D = 150\,\text{mm}$

$\text{Re}_D = 1.9 \text{x} 10^4 \sim 2.5 \text{x} 10^5$

$U_i = \dfrac{\text{Re}_{Di}\, \mu}{\rho\, D}$

2.2 Flow Analysis Setting

For turbulence model, the RNG $k - \varepsilon$ is employed in this case. It is one of the $k - \varepsilon$ variants of RANS and 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 SIMPLEC pressure correction procedure is used to handle the pressure-velocity coupling. The scheme is fully implicit in time and of second order and small time steps are used in order to minimize the discretization error.

In the present case, velocity is set in the inlet boundary condition and pressure is specified at the pipe outlet. No-slip boundary conditions are applied to the surface of bluff body as well as to the top and the bottom of the domain. The velocity inlet is assumed as constant profile and for turbulent quantities, the hydraulic diameter (D_h) and turbulent intensity (I) are used. The turbulent intensity (I) is obtained from the relationship of

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

2.3 Objective Functions

A high performance vortex shedder should be ensure to generate strong and stable vortices with low power loss whose shedding frequencies varies linearly with flow rate over a wide range of Reynold number. The linearity of Strouhal number, minimum pressure loss and drag coefficient are specified as the objectives in this study because these are important factors for a good vortex shedder [22]. Here, the linearity of the Strouhal number is assessed by error percentage (e). The error e is defined as

$$e = \frac{1}{N} \sum_{i=1}^{N} \frac{\left| St_i - \overline{St} \right|}{\overline{St}} \tag{2}$$

The Strouhal number of a vortex flowmeter is expressed as:

$$St_i = \frac{f\,d}{U_i} \tag{3}$$

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

For pressure loss, the resistance coefficients, K-factor, is checked and it is described by as follows.

$$K_i = (P_1 - P_2)/0.5\rho U_i^2 \tag{4}$$

In this equation, P_1 and P_2 are static pressure at the upstream and downstream control points respectively.

And, the coefficient of drag (C_d) for bluff body is also evaluated using the following relationship:

$$C_{d_i} = \frac{Drag\ force}{\frac{1}{2}\rho U_i^2 A_p} \qquad (5)$$

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

Here, the mean values are the average of specific parameters for the tested range of Reynold number   .

$$Mean\_X = \frac{\sum\limits_{i=1}^{N} X}{N} \qquad (6)$$

3. Integration Procedure for Optimization

ISIGHT is a Process Integration and Design Optimization (PIDO) software framework, which enables various applications to be easily integrated. With iSIGHT flexible simulation process flows can be created to automate the exploration of design alternatives and identification of optimal performance parameters.

In the present work, the optimization has been achieved using a combination of three tools: GAMBIT creating the geometry and meshing, FLUENT analyzing the fluid flow and iSIGHT driving the workflow. Additionally Microsoft Excel spreadsheets are used to define the input and to check the response for post-processing **Fig.1** is the Automated Design-Evaluate-Redesign Cycle combined the GAMBIT and FLUENT with iSIGHT. The components of this cycle are explained in the later sections.
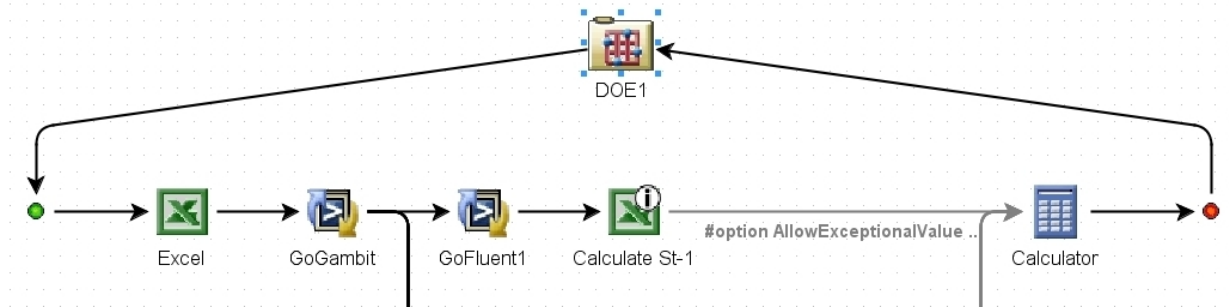

Fig.1 iSIGHT Automated Design-Evaluate-Redesign Cycle

3.1 'Excel' Component

An Excel Component is added to the simulation flow process in order to execute the activities that take input parameters, perform some functions external to iSIGHT, and provide new values to output parameters. After specifying the workbook that the component will use, name the cells or ranges as input and output parameters. These output parameters go to the specific components in the form of input parameters. The following figure is the input-output parameters for Excel component. As shown in **Fig.2**, the bluff body diameter and slit ratio are the designed input parameters intend to optimize in the current work and Excel component outputs the needed values for next executive components, GAMBIT and FLUENT.
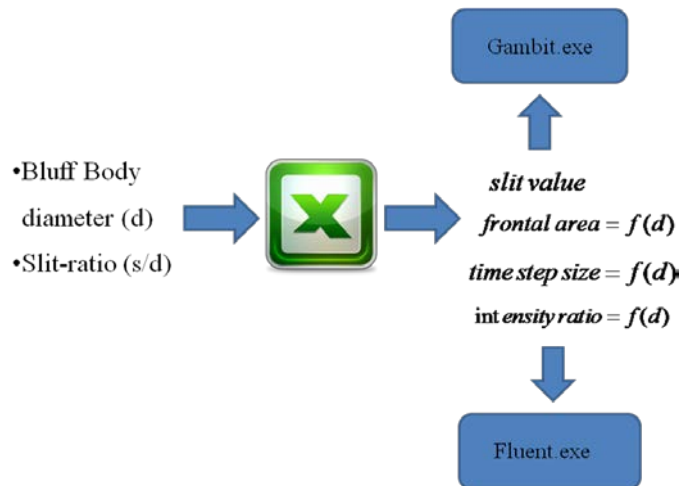

Fig.2 Input – Output parameters for Excel component

3.2 'Simcode' Component

This section demonstrates how to use the 'Simcode' component to build a simulation process flow in the iSIGHT Design Gateway with an existing executable and input and output template files. The 'Simcode' component executes an external program within the iSIGHT model. The program's input is one or more files. Parameter values are written into the files so as to allow the model to vary the inputs. Similarly, the program's output consists of one or more files. Values are read from these files and stored in parameters so they can be used in other parts of the model. ISIGHT will be running this program many times with different values of several input parameters. Therefore, it is needed to tell the iSIGHT how to put the new values into the input files so that iSIGHT can evaluate them.

In this study, two 'Simcode' components are added; the first one is for creating and meshing the geometry and another one is for fluid flow analysis. Here, the bluff body diameter (d) and slit value (s) results from the previous 'Excel' component is used as input parameters for 'simcode 1' component (see **Fig.3)** and generates the meshing file after reading the input file describing the vortex shedder geometry. **Fig.4** shows the input – output files for 'simcode 2' named FLUENT executing the fluid flow by the flow analysis setting described in previous section (2.2).
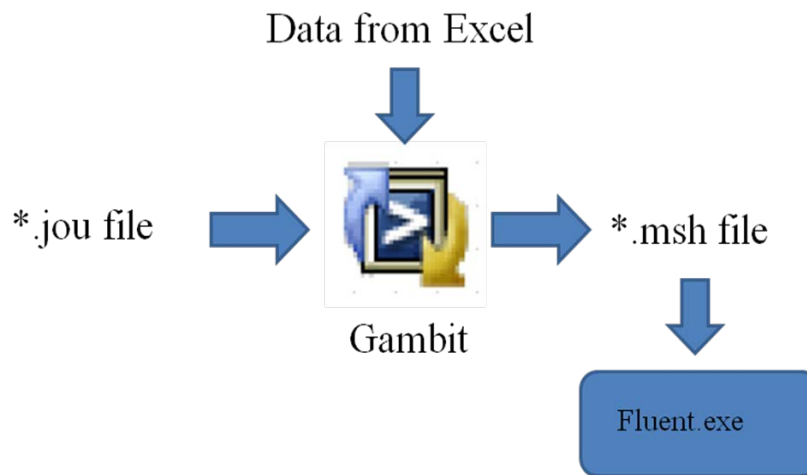

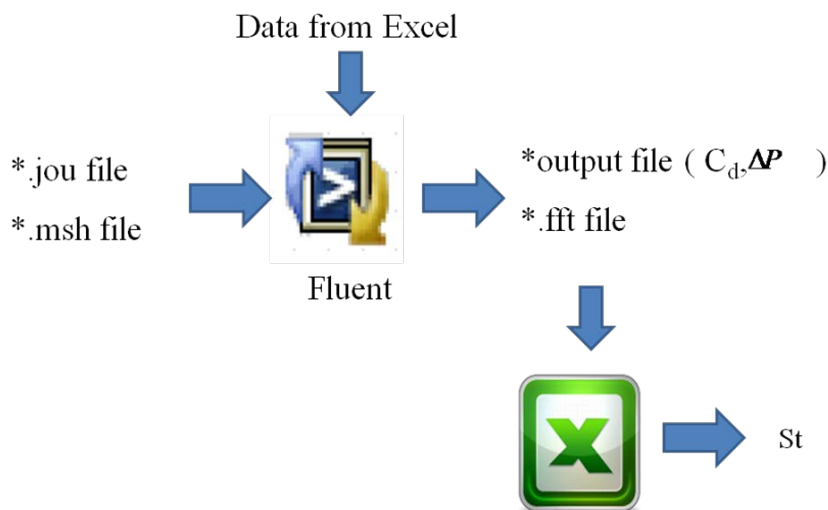
Fig.3 Input – Output Files for Simcode-1

Fig.4 Input – Output Files for Simcode-2

3.3 Design Algorithm Setup

Design optimization is the process whereby a selected set of input design variables is varied automatically by an algorithm in order to achieve more desired outputs. In the present analysis, the DOE driver is used and the applied technique is FULL FACTORIAL DOE technique. This technique enables to specify the number of levels for each factor and all combinations of all factors at all levels are studied. At present work, three design levels are specified for variable "d" and five design levels for variable "s/d" respectively. As all possible factor interactions can be evaluated in this DOE technique, the total number of program running time is 15 times. The design matrix generated for specified design points in design variables can be seen in **Fig.5**.
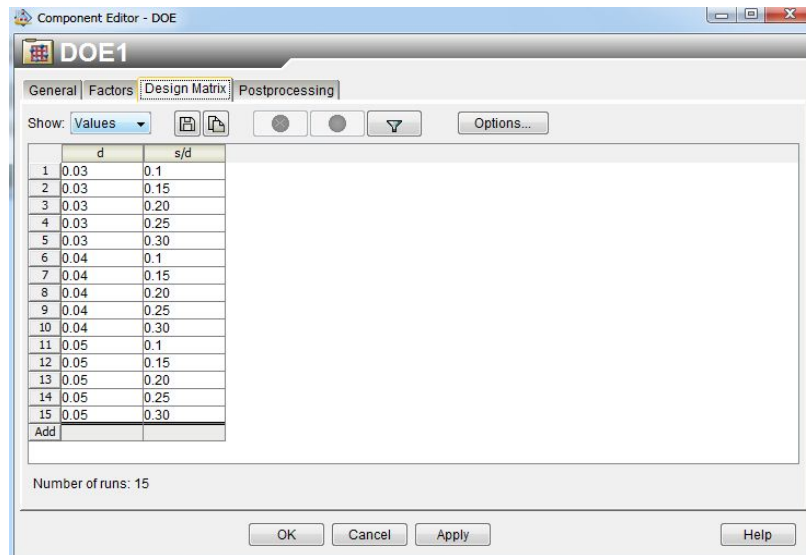
Fig.5 Design Matrix generated

4. Design Results

4.1 Grid Independence Study

For an unsteady flow simulation, the results should be independence on the grid generation in order to achieve the accurate ones. It has found that Mesh III generates the data in well agreement with the experiment and no significant changes for further refinement Mesh IV. Hence, Mesh III is employed for optimization. **Fig.6** shows the grid refinement test for this optimization and it is conducted for a typical case of d=30mm and s/d=0.1 circular with silt at $Re_D = 1.9x10^4$.
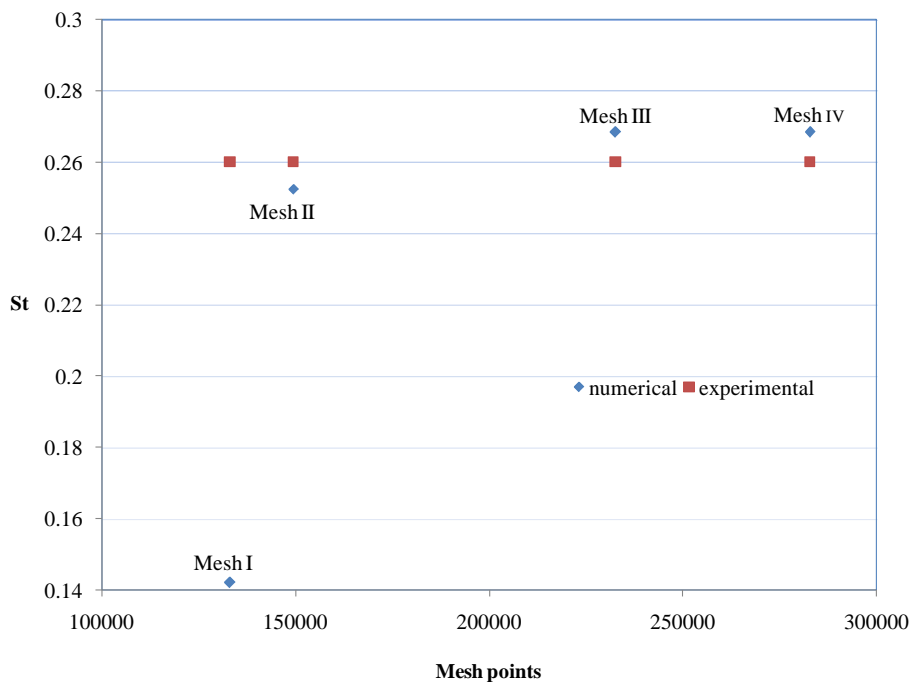

Fig.6 Grid refinement test for optimization

4.2 Results generated by Automated Design Cycle

Normally, the optimization has been processed individually. The geometry is created by pre-processor GAMBIT, the physical boundary condition is specified and mesh file is exported to solve the problem. Then, the flow condition around bluff body is analyzed by computational fluid dynamic code, FLUENT and choose the best design point corresponds to the design objective. In this optimization, however, the combination of GAMBIT, FLUENT and iSIGHT is used with DOE driver. This combination method substitutes the data automatically according to the input parameters. Furthermore, the iSIGHT can keep the result history for all design matrixes. As the results for each design point is saved in one place, assessment and comparison can be made easily. The DOE results history generated by iSIGHT's automated design cycle is described in **Fig.7**. The green coloured tick in the first column means
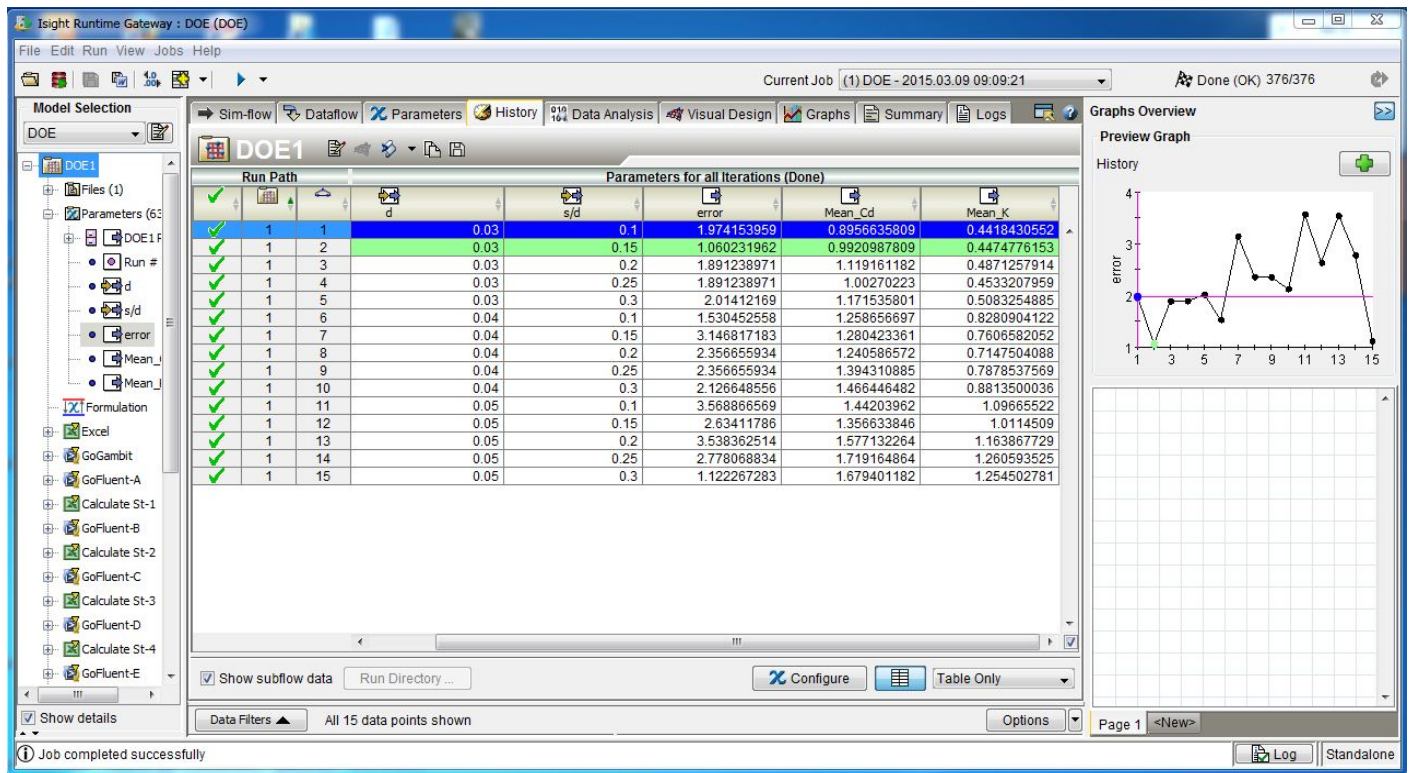
that this work has been run successfully.

Fig.7 DOE result history generated for Tri-semi circular cylinder

Fig.8 and **Fig.9** illustrates the Pareto Graphs of the effect of design parameters on objective functions for circular with slit and tri-semi cylinder vortex shedder respectively. By these graphs, the Pareto contribution ratio of parameter "s/d" to e and C_d is around 30% while that of the parameter "d" is over 60% (difference in percentage is just about double). But, the contribution of "d" is 6 times more than the contribution of "s/d" to K-value (60% Vs 10%). So, it can be said that the variable "d" affects on drag coefficient "C_d" and error of linearity "e" as much as the variable "s/d" although the variable "d" much more affects on K-factor than the variable "s/d". On the other hand, for error of linearity and drag coefficient, both variables influence on the objective functions and, for pressure loss coefficient, variable "d" is more important than variable "s/d". In other words, shedder diameter and slit ratio are important to generate linear Strouhal number and minimum drag coefficient whilst diameter is more influence on pressure loss to be low than slit ratio does.
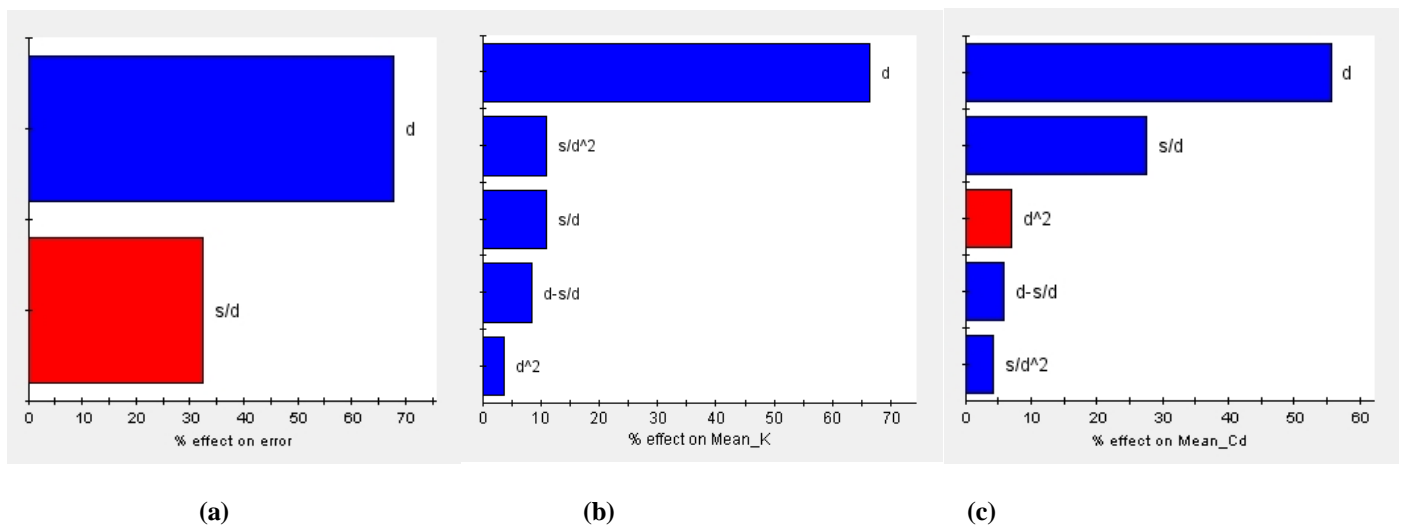

(a) (b) (c)

Fig. 8 Effect of design parameters on **(a)** Error of Linearity **(b)** Mean_K **(c)** Mean_C_d (Tri-Semi Cylinder)

Contour for the relationship between objective functions and design parameters are presented in **Fig.10** and **Fig.11**. **Fig.10(a)** shows the contour of error versus d, s/d and it is found that the most linearity in Strouhal number is generated by the shedders having 30mm diameter with 0.15 slit ratio and 50mm diameter with 0.30 slit ratio. It means that these design dimensions give the least error in linearity. By **Fig.10(b)**, shedder of d=30mm generates low pressure loss for all s/d. For drag coefficient, **Fig.10(c)** presents that d=30mm s/d=0.1 is the best for minimum drag coefficient.

155

As can be seen in **Fig.11**, d=30 mm having s/d=0.1 and d=50mm with s/d=0.25 is the lowest for error percentage. In **Fig.11(b)**, it is found that the (30~35mm) diameter shedder with (0.1~0.3) ranges of slit ratio have the low pressure loss. And **Fig.11(c)** demonstrates that the same diameter shedder with (0.1~0.2) ranges of slit ratio give minimum drag coefficient.

(a)   (b)   (c)
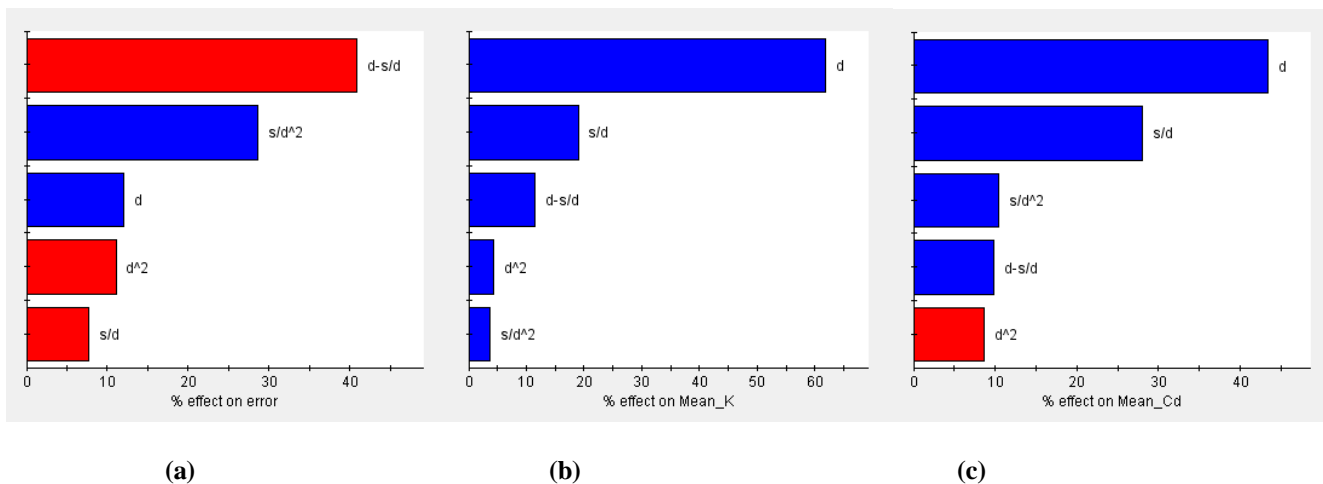
Fig.9 Pareto Plot for **(a)** Response Error **(b)** Response Mean_K **(c)** Response Mean_C_d (Circular with slit)

(a)   (b)   (c)
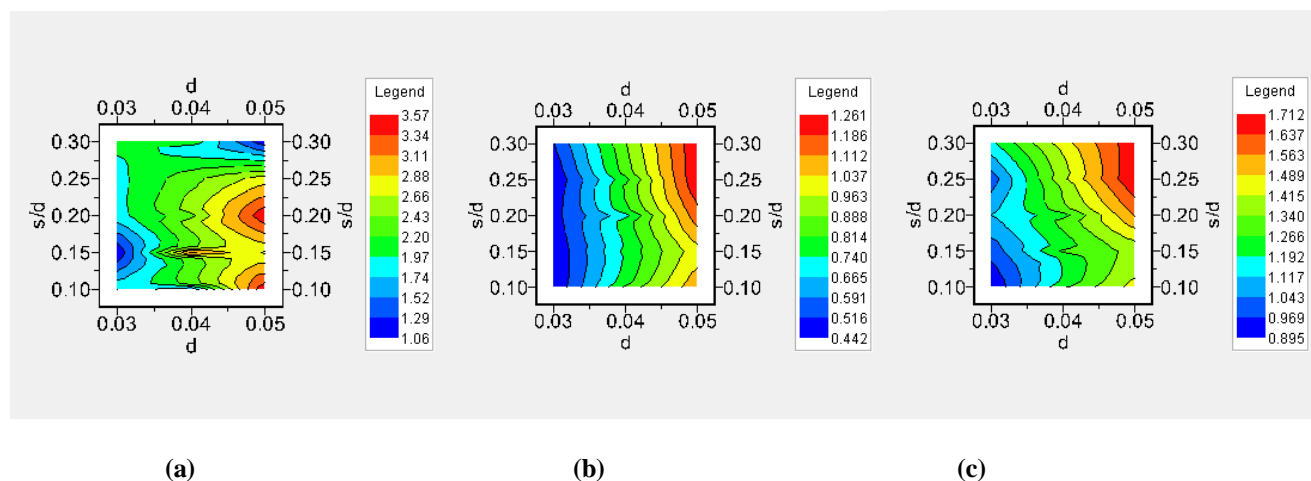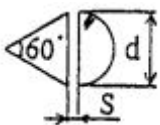
Fig.10 Contour of **(a)** Error (%) Vs. d, s/d **(b)** Mean_K Vs. d, s/d **(c)** Mean_C_d Vs. d, s/d (Tri-Semi Cylinder)

(a)   (b)   (c)
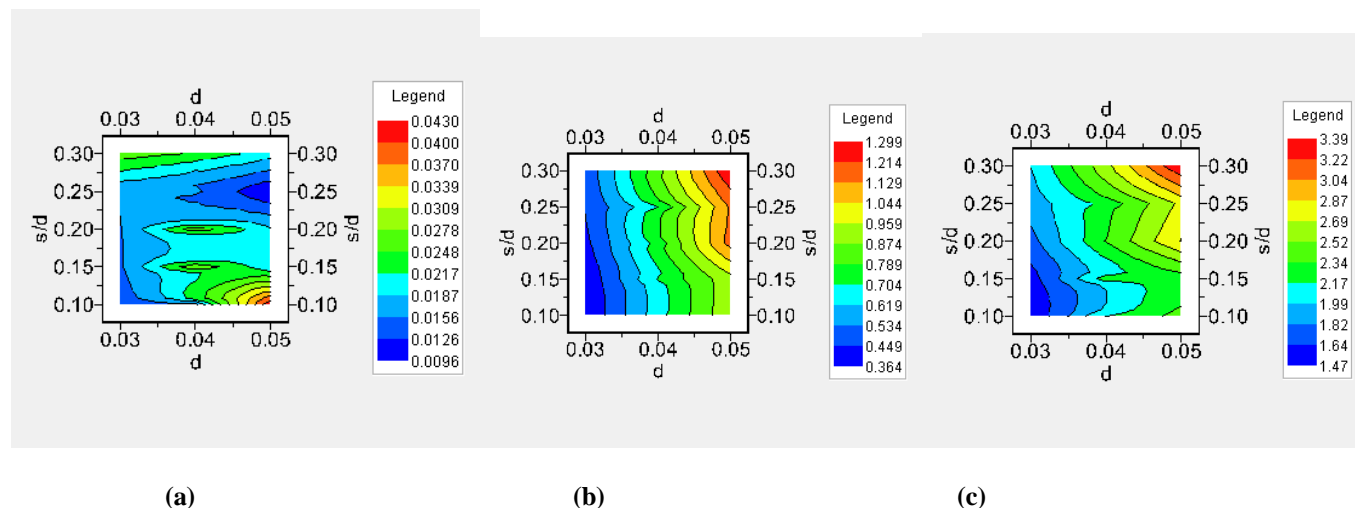
Fig.11 Contour of **(a)** Error (%) Vs. d, s/d **(b)** Mean_K Vs. d, s/d **(c)** Mean_C_d Vs. d, s/d (Circular with slit)

To be a good performance vortex shedder, it is essential to generate the strong and stable vortex under the linearity in Strouhal number and low pressure loss. By these properties, it can be found that the circular cylinder with slit vortex shedder in the diameter of 30mm and slit ratio of 0.1 and tri-semi cylinder shedder in the diameter of (30~35) mm diameter with 0.15 slit ratio are the optimized designs. The optimized designs by present numerical integration method are compared with experimental results in Table 2.

Table.2. Optimized Shedder Dimensions

Shedder	Numerical		Experimental [11]	
	d (mm)	s/d	d (mm)	s/d
	30	0.1	30	0.1
	30~35	0.15	40	0.15

5. Conclusion

In the present work, the integration of GAMBIT and FLUENT to iSIGHT has been applied in order to get the optimal design for vortex shedder by inputting the data itself. In optimization process, the simulation has been performed for unsteady flow in the turbulent Reynold number, ranged from 1.9×10^4 to 2.5×10^5. RNG $k-\varepsilon$ turbulent model is used for simulation worked in FLUENT. The two vortex shedders in the form of circular with slit and triangular- semi circular cylinder are used as the optimized objects.

The essential properties for a good performance vortex shedder have been considered for optimal design. According to these essential properties, it has found that the circular cylinder with slit in the diameter of 30mm and slit ratio of 0.1 and tri-semi cylinder shedder in the diameter of (30~35) mm diameter with 0.15 slit ratio are the optimized designs. And it is found that these optimal results generated by DOE automated design cycle agree with the experimental results.

The design solution has been achieved by the FULL FACTORIAL DOE designing method with reduced human efforts. However, the DOE technique results the optimal values that includes in the specified level. In the future, we will adopt the Optimization technique to further optimization design in which current design solution will be used as the first generation to achieve the more précised optimal designing solution. Therefore, it is recommended that this integration method is useful in practical engineering application and will be an efficient method to develop the new designs for vortex shedder with less design cycle time and low manufacturing cost eliminating the man-made-errors.

References

[1] A. K. EL WAHED and J. L. SPROSTON, 1991,"The influence of shedder shape on the performance of the electrostatic vortex flowmeter," Flow Meas. Instrum., 2 July,

[2] M. W. JOHNSON J. L. SPROSTON A. K. EL WAHED, 1993,"Numerical study of vortex shedding from different shaped bluff bodies," Flow Meas. Instrum., 4, 4,

[3] Y. YAN J. COULTHARD 1993,"Comparisons of different bluff bodies in vortex wake transit time measurements," Flow Meas. Instrum., 4, 4,

[4] Olsen J.F. and Rajagopalan S., 2000,"Vortex shedding behind modified circular cylinders," Journal of Wind Engineering and Industrial Aerodynamics, 86, 55-63.

[5] et al. B K Gandhi, 2004,"Effect of Bluff Body Shape on Vortex Flowmeter Performance," Idian Journal of Engineering & Material Sciences, 11, 378-384.

[6] TAO ZHANG DAN-DAN ZHENG, 2011,"Comparison Study on Flow Fields in Vortex Flow Sensors with Circular Cylinder and Trapezoidal Cylinder Bluff Bodies," Proceedings of the 2011 International Conference on Machine Learning and Cybernetics,,

[7] Amit Agrawal A. Venugopal, S.V. Prabhu, 2011,"Influence of blockage and shape of a bluff body on the performance of vortex flowmeter with wall pressure measurement," Measurement, 44, 954-964.

[8] A. Venugopal Lavish Ordia, Amit Agrawal, S. V. Prabhu, 2013,"Influence of After Body Shape on the Performance of Blunt Shaped Bodies as Vortex Shedders," World Academy of Science, Engineering and Technology, International Journal of Mechanical, Industrial Science and Engineering, 7, 10,

[9] LIZhiyong SUN Zhiqiang, JIANG Yun,ZHOU Hongliang, 2013,"Influence of bluf f body shape on wall pressure distribution in vortex flowmeter ," J. Cent. South Univ . , 20, 724-729.

[10] C. C. YANG J. J. MIAU, J. H. CHOU , K. R. LEE, 1993,"A T-shaped vortex shedder for a vortex flowmeter," Flow Meas. Instrum., 4, 4,

[11] Igarashi T., 1999,"Flow resistance and strouhal number of a vortex shedder in a circular pipe," JSME International Journal, 42, 4, 586-595.

[12] M. Takamoto & Y. Terao, 1999,"STANDARDIZATION OF VORTEX SHEDDING FLOWMETER," 4th International

Symposium on Fluid Flow Measurement,

[13] Jiegang Peng, Xin Fu and Ying Chen, 2004,"Flow measurement by a new type vortex flowmeter of dual triangulate bluff body," Sensors and Actuators, 115, 53–59.

[14] Xin Fu Jiegang Peng, Ying Chen, 2008,"Experimental investigations of Strouhal number for flows past dual triangulate bluff bodies," Flow Measurement and Instrumentation 19, 350-357.

[15] Amit Agrawal A. Venugopal, S.V. Prabhu, 2010,"Influence of blockage and upstream disturbances on the performance of a vortex flowmeter with a trapezoidal bluff body," Measurement, 43, 603-616.

[16] 2003,"iSIGHT User's Guide", version 8.0,

[17] 2009,"iSIGHT 4.0 Getting Started Guide,"

[18] XU Dian   YIN Bo, AN Yi-ran, CHEN Yao-song, 2008,"Aerodynamic optimization of 3D wing based on iSIGHT," Applied Mathematics and Mechanics (English Edition), 29, 5,

[19] Rong Mo Wei Wang, Zhiyong Chang, Huibin Sun, 2010,"Aerodynamic optimization designing of Airfoil based on Isight," IEEE,

[20] Rong Mo Wei Wang , Yan Zhang, 2011,"Multi-objective aerodynamic optimization design method of compressor rotor based on Isight," Advanced in Control Engineering and Information Science, 15,

[21] CHEN GuoPing SUN RuJie, ZHOU Chen, ZHOU LanWei & JIANG JinHui, 2013,"Multidisciplinary design optimization of adaptive wing leading edge," SCIENCE CHINA Technological Sciences, 56, 7,

[22] Su Myat Nyein and He Xu, 2015,"Numerical Study of Important Factors for a Vortex Shedder using Automated Design Cycle," International Journal of Fluid Machinery and Systems, 8, 2,