A practical verification and validation approach for Computational Wind Engineering simulations using an experimental design technique

Réamonn Mac Réamoinn [a], DongHun Yeo [b]

[a] *Arup, Dublin, Ireland, r.macreamoinn@gmail.com*
[b] *National Institute of Standards and Technology, Gaithersburg, Maryland, U.S.A., donghun.yeo@nist.gov*

SUMMARY:
Understanding the sensitivity of computational wind engineering (CWE) solutions to simulation parameters facilitates the development of useful solutions. The traditional approach adopted in other fields through verification and validation (V&V) with well-defined benchmarks is not always possible in CWE. Complex flow phenomena in even simple flow conditions are not readily defined analytically or measured experimentally. This study aims to evaluate the sensitivity of CWE solutions to various simulation parameters with a view to identifying the optimal simulation configuration yielding more reliable results with the least computational effort. To this end the study uses the experimental design technique and seeks to verify the optimal size of the computational domain and level of mesh refinement.

Keywords: Computational Wind Engineering, Experimental Design Technique, Orthogonal Fractional Factorial Design, Square Cylinder Flow, Vortex Shedding

1. INTRODUCTION

In Computational Wind Engineering (CWE), a subset of computational fluid dynamics (CFD), results of simulations of the same flow condition using different model configurations can vary widely. This undermines user confidence in CWE solutions. If CWE is to be used as a tool for the design of special structures, such as tall buildings or long span bridges, in conjunction with or in place of wind tunnel testing, it is necessary to improve its credibility.

To facilitate the development of more accurate and reliable simulations it is necessary to understand the sensitivity of CWE solutions to simulation parameters. In other fields, this can be achieved through verification and validation (V&V) with well-defined benchmarks (Oberkampf & Trucano, 2008). However, in CWE complex flow phenomena can occur under relatively simple flow conditions. These phenomena, such as boundary layer separation and signature turbulence, are not readily defined analytically nor easily measured experimentally. Roache (1997) recognized that "useful *a priori* estimation is not possible for nontrivial fluid mechanics problems". Therefore, it is not always possible to adopt the same approach for CWE.

This study aims to evaluate the sensitivity of CWE solutions to various simulation parameters with a view to identifying the optimal simulation configuration yielding more reliable results with the least computational effort. In any simulation or experiment, the accuracy of the solution is improved through the reduction of error. The confidence in the reliability of the solution is

related to the quantifying the error and the identification of its sources.
The convergence of a solution is linked to the reduction of the numerical errors associated with simulation parameters. This study proposes a practical method that helps evaluate if a solution can be considered as being independent of simulation parameters. To this end it develops a V&V approach consisting of an experimental design technique for assessing the convergence of a solution. This technique makes it possible to identify the extent to which simulation results are sensitive to parameter values, evaluate the influence of single parameters and their interaction on the accuracy of the solution, and provide the optimal set of parameter values that minimize numerical errors while using typical computational resources.

As an application of the V&V approach developed in this study, the high Reynolds number flow around a square cylinder is examined. In spite of its geometric simplicity, complex flow phenomena can develop, such as vortex shedding and boundary layer separation. However, unlike in the case of a circular cylinder flow, the square cylinder flow has fixed points of separation, which implies that the flow could be assumed to be Reynold's number independent. This fundamental problem in bluff body aerodynamics has been extensively researched in both numerical simulations (Dahl, 2014, Franke et al., 1990, Lee and Bienkiewicz, 1998, Shimada and Isihara, 2002, Tamura and Yoshiyuki, 2003, Tain et al., 2012) and wind tunnel tests (Bearman and Trueman, 1972, Bearman and Obasaju, 1982, Durao et al., 1988, Lee, 1974, Lyn and Rodi, 1993, Lyn et al., 1995). Therefore, it is considered a suitable case for assessing the effectiveness of this approach.

2. EXPERIMENTAL DESIGN TECHNIQUE

Experimental design is a statistical tool that makes it possible to achieve experiment objectives by planning efficiently experimental procedures in which changes in one or more process parameters affect one or more output responses. Once the objective of a study and the process factors are determined, well-established experimental design techniques economically maximize the amount of information obtainable from a minimized amount of experimental efforts.

A traditional and common approach to experimental design involves only changing a single input parameter or factor between runs, which is known as the *one factor-at-a-time* (1FAT) *design*. As noted by Box et al. (1978), "the method provides an estimate of the effect of a single variable at selected *fixed* conditions of the other variables." It assumes the effect would be the same irrespective of the condition of the other variables and therefore, it neglects any interaction between variables. Given that 1FAT design approach ignores interactions, its estimates are biased and lack of precision if the interaction between any parameters is significant.

It is possible to design experiments to take account of interaction between parameters through orthogonality. In orthogonal designs, multiple parameters are changed between experiments. However, through controlling the adjustments to the parameters, it is possible to ensure the interaction effects between parameters balance. As a result, orthogonal design facilitates more accurate estimates of main and interaction effects of parameters when compared with more widely used approaches. When all the possible combinations of parameters are considered, it is known as a *full factorial design*. This takes account of all the interactions between parameters while producing unbiased estimates. However, it does demand a large number of experiments be undertaken. In a *fractional factorial design*, a smaller appropriately chosen subset is adopted. It takes advantage of redundancy within the full factorial design to produce equivalent estimates while significantly reducing the number of experiments undertaken without compromising the capacity to discrimination between parameters.

In this study, it is proposed to adopt a modified orthogonal fractional factorial approach to evaluate the sensitivity of the CWE solution to a variety of parameters in minimized but well selected simulation cases. The modification involves the inclusion of an additional case, which has an intermediate value of parameters between their extreme values in the traditional factorial approach. The addition of the 'center' case makes it possible to investigate the influence of parameters on simulation results within their extreme bounds, and to assess convergence of the results as the parameter values are adjusted.

3. NUMERICAL ARRANGEMENT

This study examines 2D flow around a square cylinder. The characteristic dimension, H, of the square cylinder is taken as 0.04 m and the freestream velocity, U_∞, of the flow is taken as 8.2 m/s. The flow medium is air with a density ρ = 1.20 kg/m^3 and a kinematic viscosity, υ = 1.51x10^{-5} m^2/s. The Reynolds number is therefore 20000. The turbulence effects in the flow are modelled using the Spalart-Allmaras turbulence model. A turbulence intensity I = 2 % is assumed. The simulations are carried out using the open source CFD package, OpenFOAM (The OpenFOAM Foundation, 2016). In all cases, a steady state solution was used as an initial condition. The steady state and transient simulations were solved using simpleFoam and pimpleFoam, respectively.

The initial and boundary conditions are specified on the domain boundaries. The inflow condition is prescribed as a uniform freestream velocity of 8.2 m/s normal to the incoming flow boundary surface. The turbulent viscosity, υ, is defined as 5.62x10^{-4} m^2/s on the inflow boundary. The dynamic pressure was prescribed as zero on the outlet boundary. A 'no slip' condition was applied to the square cylinder surface and wall functions were used to approximate the flow near the cylinder surfaces. A 'slip' condition was applied to the top and bottom boundaries, while the front and back boundaries were defined as 'empty' condition to maintain the 2D flow conditions.

The configuration of the mesh within the computational domain is illustrated in Figure 1 and 2. The background mesh, which has a cell size of H/8, consists of a uniform orthogonal hexagonal mesh. The mesh refinement occurs locally to the square cylinder. The mesh layers with H/8 of the cylinder surface have a cell height of H/64. The mesh transitions from H/8 to H/64 within the refinement zone. Depending on the simulation case, the cell widths along the cylinder surface range between H/16 and H/64. The transitioning in the cell widths also occurs within the refinement zone.

This study investigates the influence of four simulation parameters on CWE solutions. It is acknowledged that a wide variety of parameters could be selected including, but not limited to the time-step, mesh type, mesh expansion ratio, numerical schemes and solvers. However, this study is most concerned with domain configuration and mesh refinement as it is considered that these parameters will have the most significant influence on the result. Therefore, the following four parameters are investigated in this study:

- x1: Upstream length;
- x2: Downstream length;
- x3: Cross-stream width;
- x4: Mesh refinement.

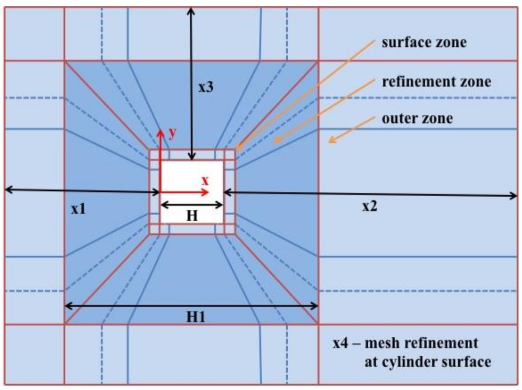

Figure 1. Extent of computational domain

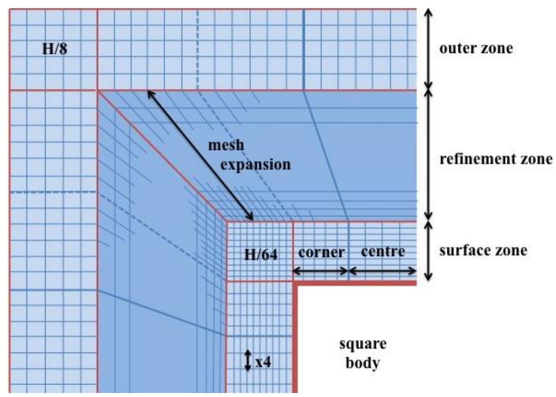

Figure 2. Mesh refinement zones within computational domain

Table 1. Simulation cases configuration based on modified half fractional factorial design approach

Parameters	Upstream	Downstream	Cross-stream	Mesh Size
-	7.5H	15H	7.5H	H/16
o	15H	30H	15H	H/32
+	22.5H	45H	22.5H	H/64
Case	x1	x2	x3	x4
s1c1	-	-	-	-
s1c2	+	-	-	+
s1c3	-	+	-	+
s1c4	+	+	-	-
s1c5	-	-	+	+
s1c6	+	-	+	-
s1c7	-	+	+	-
s1c8	+	+	+	+
s1c9	o	o	o	o

Given that the experimental design technique considers four different parameters and each parameter is assigned a lower (-) or upper (+) value (e.g., 7.5H or 22.5H for x1 parameter), at least 8 simulations are required. However, an additional simulation, or target case, is added to investigate intermediate (o) values (e.g., 15H for x1 parameter) in-between the extremes considered in the factorial design approach. The configurations of the 9 simulations, which are investigated in this study, are outlined in Table 1 below.

4. RESULTS

Visual inspection of the flow fields for each of the simulations reveal the characteristic alternating vortex shedding phenomenon originating on the upper and lower surfaces of the cylinder before progressing further downstream. The velocity and pressure fields with overlapping vorticity contours are shown in Figures 3 and 4, respectively.

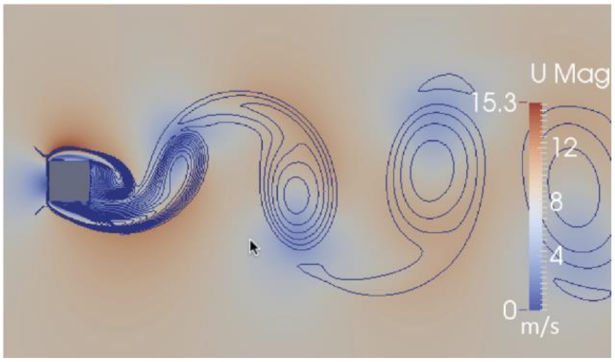

Figure 3. Instantaneous velocity field with vorticity contours (s1c9)

Figure 4. Instantaneous pressure field with vorticity contours (s1c9)

Table 2. Simulation flow parameter results for all 9 simulations

Case	Inflow velocity (P1) U_{inflow}	Drag coeff. (P2) C_D	Lift coeff. (P3) $C_{L,rms}$	Base press. (P5) $\lvert C_{p,base}\rvert$	Strouhal no. (P6) St	Cross-stream (P7) U_{cross}	Recirculation zone (P8) L_{zone}	Downstream velocity (P9) U_{down}
	[-]	[-]	[-]	[-]	[-]	[-]	[-]	[-]
s1c1	1.004	2.110	1.460	1.396	0.145	0.249	0.506	0.983
s1c2	1.011	2.138	1.482	1.415	0.135	0.252	0.495	0.946
s1c3	1.004	2.107	1.451	1.382	0.145	0.260	0.513	0.958
s1c4	1.012	2.145	1.499	1.437	0.134	0.241	0.491	0.963
s1c5	1.000	1.999	1.388	1.260	0.146	0.264	0.527	0.953
s1c6	1.005	2.050	1.446	1.331	0.135	0.234	0.481	0.945
s1c7	1.000	2.008	1.367	1.278	0.146	0.248	0.495	0.946
s1c8	1.005	2.005	1.408	1.322	0.135	0.253	0.509	0.926
s1c9	1.006	2.066	1.434	1.337	0.135	0.248	0.500	0.943

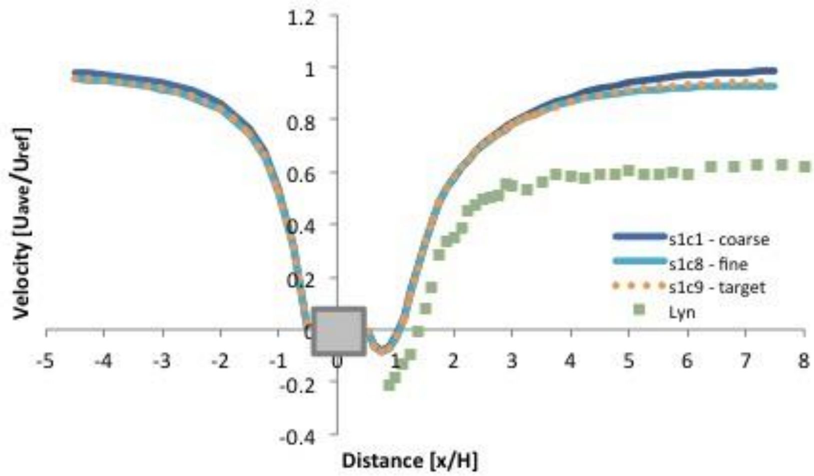

Figure 5. Time-averaged wind velocity along centerline, U_{ave}/U_{ref}

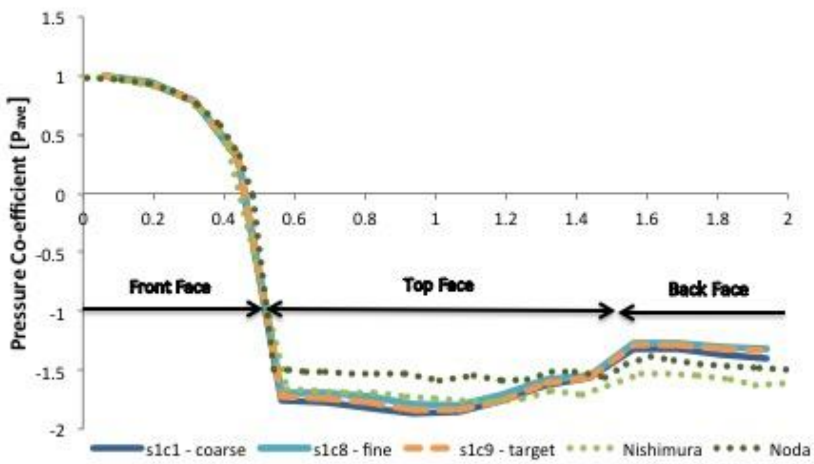

Figure 6. Time-averaged pressure coefficient, C_p, around cylinder surface

The results for a variety of flow parameters have been collected for each of the simulations. These include parameters related to the flow field such as the time-averaged inflow velocity, the time-averaged maximum cross-stream velocity, the length of the recirculation zone and the time-averaged velocity 7.5H downstream of the rear of the cylinder along the centreline of the domain. They also include parameters related to the pressure field, including the magnitude of the base pressure coefficient at the rear of the cylinder as well as the drag and lift coefficients. It should be noted that both velocity and pressure field parameters have been normalized relative to the stagnation pressure. In addition, the Strouhal number was also calculated. These results are summarized in Table 2 above.

5. DISCUSSION

5.1. Overview

Upon initial visual investigation, all simulations appear to give good results. It is apparent that free shear layers form at the front corners of the cylinders. Due to instability within these layers, the shear layers interact and roll-up. Ultimately, this leads to the formation of vortices at the rear of the cylinder that are shed periodically downstream.

However, closer investigation does highlight a few areas of concern. For instance, the maximum magnitude of velocity, as shown in Figure 3, is significantly higher than expected. It is almost double the inflow velocity rather than between 20 % and 50 % higher. Furthermore, the velocity downstream of the cylinder along the domain makes an almost full recovery within 8H of the cylinder. Moreover, the vorticity contours follow the mesh boundaries as they extend upstream from the front face of the cylinder. It is clear that the mesh construction influences the flow field.

5.2. Experimental Design Technique

5.2.1. Velocity Field

Using the contrast of coefficients method outlined by Box et al. (1978), it is possible to identify the main and interaction effects of the different parameters. The main effects are summarized in Table 3 and illustrated in Figures 7 to 10. The interaction effects were found to be less significant and, therefore, have been omitted except for the recirculation zone length (Fig. 11).

Table 3. Main effects of simulation parameters on velocity field parameters

Parameter		Average	Main Effects			
			x1 (Upst. length)	x2 (Downst. length)	x3 (Crossst. length)	x4 (Mesh refin.)
P1	U_{inflow}	1.008	0.006 0.6 %	0.000 0.0 %	-0.005 -0.5 %	0.000 0.0 %
P7	U_{cross}	0.251	-0.010 **4.1 %**	0.001 0.3 %	-0.001 -0.3 %	0.014 **5.7 %**
P8	L_{zone}	0.501	-0.016 **-3.2 %**	0.000 -0.1 %	0.002 0.4 %	0.018 **3.5 %**
P9	U_{down}	0.962	-0.015 **-3.2 %**	-0.008 -0.9 %	-0.020 **-2.1 %**	-0.013 **-1.4 %**

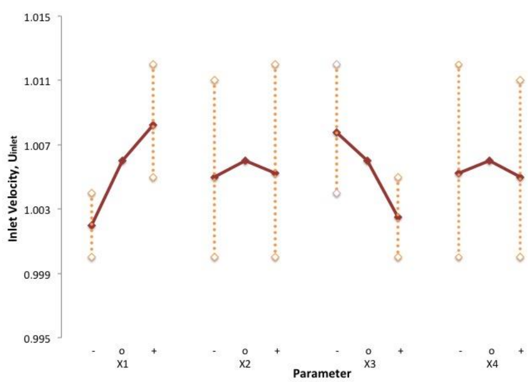

Figure 7. Inflow velocity (P1)

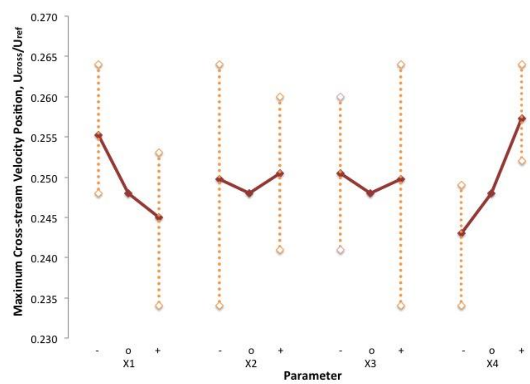

Figure 8. Cross-stream velocity (P7)

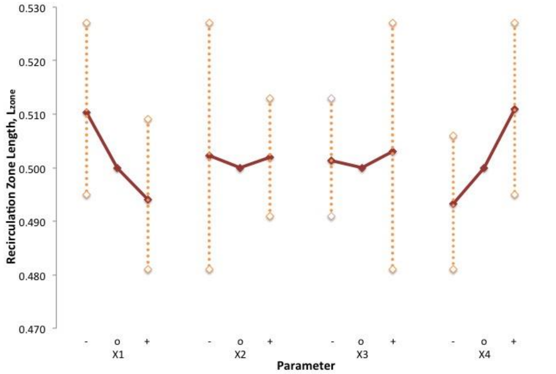

Figure 9. Recirculation zone length (P8)

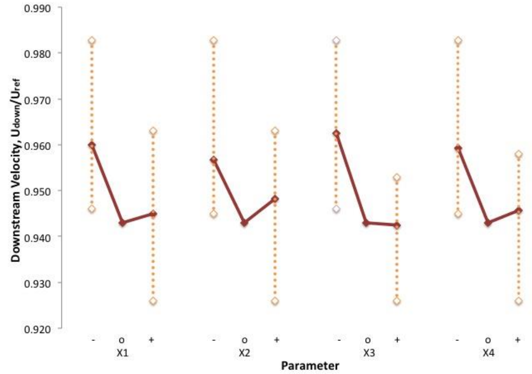

Figure 10. Downstream velocity (P9)

As shown in Fig. 7, the inflow velocity (P1) is influenced by the upstream length (X1) and the cross-stream length (X3). The short distance from the inflow boundary to the cylinder decreases the velocity at the boundary. However, the short distance from the top/bottom boundary to the cylinder increases the velocity, which is induced by funnelling effect. Note that the blockage ratios are 6.25 %, 3.2 % and 2.2 %, respectively, for the narrowest to widest domains. The upstream length and the mesh refinement are found to have a minor influence the inflow velocity.

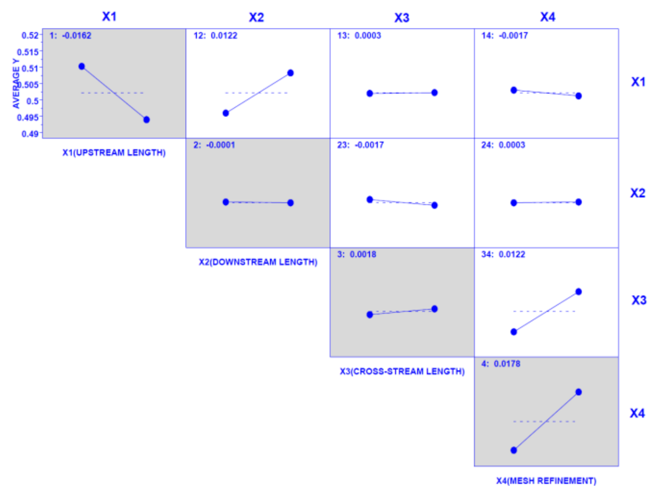

Figure 11. Interaction effects matrix: Recirculation zone length (P8)

The maximum cross-stream velocity (P7) above the middle of the cylinder's top face and recirculation zone (P8) are also influenced by upstream length and mesh refinement (Figs. 8, 9). The increase in upstream length leads to a reduction with in cross-stream velocity and recirculation zone size. While an increased mesh resolution results in an increase in cross-stream velocity and recirculation length.

Downstream velocity (P9) is dependent on all the parameters considered but, remarkably, it is least dependent on downstream length. Moreover, it is apparent from Figure 10 that an intermediate level of refinement is adequate to produce a satisfactory result.

Overall, it would appear that the velocity field is strongly influenced by the upstream width and the mesh refinement.

5.2.2. Pressure Field

The main effects are summarized in Table 4 and illustrated in Figures 12 to 14. The interaction effects were found to be less significant and therefore, have been omitted.

Table 4. Main effects of simulation parameters on pressure field parameters

Parameter		Average	Main Effects			
			x1	x2	x3	x4
P2	C_D	2.125	0.028 **1.3 %**	-0.008 -0.4 %	-0.110 **-5.2 %**	-0.016 -0.8 %
P3	$C_{L,rms}$	1.473	0.042 **2.9 %**	-0.013 -0.9 %	-0.071 **-4.8 %**	-0011 -0.7 %
P5	$\|C_{p,base}\|$	1.408	0.047 **3.4 %**	0.004 0.3 %	-0.110 **-7.8 %**	-0.016 **-1.1 %**

The cross-stream width has a strong influence over the drag and lift coefficients (P3 & P4) and the base pressure (P5). As the cross-stream width decreases, the accelerated flows above the top surface and below the bottom surface of the cylinder increase the strength of vorticity. As a result, the average drag, lift fluctuations, and pressure suction at the base increase. The upstream length has a considerable influence on those parameters. They increase with increasing upstream length.

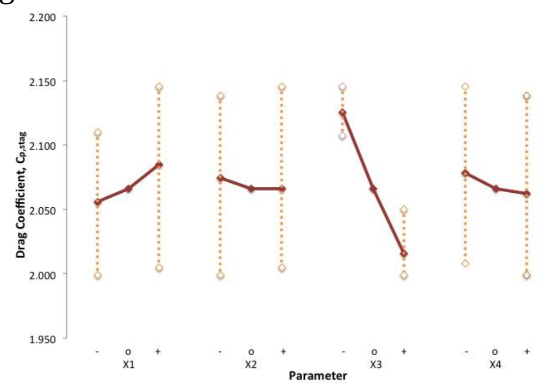

Figure 12. Time-averaged drag coefficient (P2)

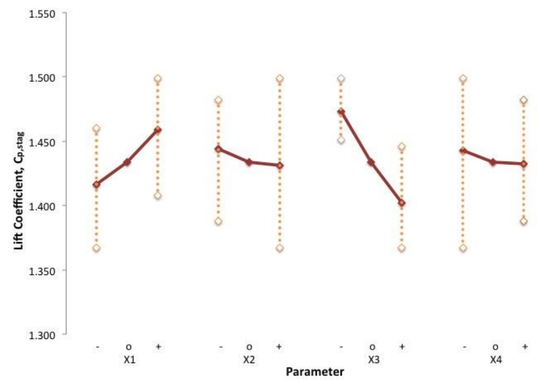

Figure 13. Root-mean-square of lift coefficient (P4)

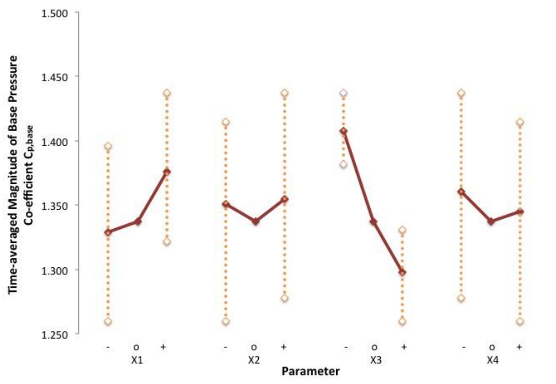

Figure 14. Time-averaged magnitude of base pressure (P5)

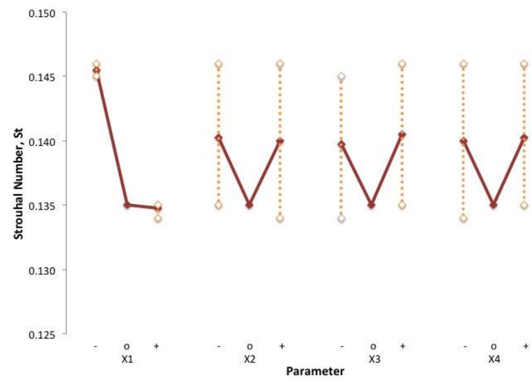

Figure 15. Strouhal number (P6)

Overall, it would appear that the pressure field is strongly influenced by the upstream length and the cross-stream width. Note that the pressure field is not sensitive to mesh refinement based on the grids employed in this study.

Table 5. Main effects of simulation parameters on Strouhal number

Parameter		Average	Main Effects			
			x1	x2	x3	x4
P6	St	0.140	-0.011 **-7.7 %**	0.000 -0.2 %	0.001 -0.5 %	0.000 -0.2 %

5.2.3. Strouhal Number

The main effects are summarized in Table 5 and illustrated in Figure 15. The Strouhal number is strongly influence by the upstream length only. As shown in Fig. 15, the intermediate upstream length (15H) is sufficient to produce a satisfactory result.

5.3. Practical Verification and Validation

The parameter values can be selected once the analysis of the sensitivity of the results pertaining to the flow or pressure field to changes in the parameter values is performed. The optimal set of parameters should be chosen to minimize the numerical errors while keeping the simulation time affordable. This is considered to be a *verification procedure* for users, though not for code developers. The simulation based on the optimal set of parameters provides the best target results in a practical manner, and can be viewed as a comparison with experimental/numerical results of the same problem. Hence, this can be considered as a *validation procedure*. Note that the verified simulation results cannot guarantee good agreement with experimental/numerical results used in the validation. The difference would not be due to numerical errors corresponding to the chosen parameters, but mainly to inappropriate modelling of the physics. If this is the case, a better model should be considered.

For this case study, to obtain reliable and efficient simulation results, the simulations for the velocity (P1, P7 - P9) and pressure/force results (P2, P3, P5) require, respectively, 15H and 15H for the upstream length (X1), 30H and 15H for the downstream length (X2), 15H and 22.5H for cross-stream length (X3), and H/64 and H/16 for the mesh size (X4). With respect to the validation of the simulation results with experimental data, Figures 5 and 6 show a comparison of the simulated velocity along the centerline and the pressure distribution on the cylinder with experimental recorded by Lyn et al. (1995), Nishimura and Tanijke (2000), and Noda and Nakayama (2003). There are large differences between experimental results and the results with reduced numerical errors corresponding to the chosen parameters. This is mainly caused by the performance of the 2D RANS turbulence model used in this study.

6. CONCLUSIONS

This study developed a practical verification and validation (V&V) approach based on the experimental design technique for computational wind engineering applications. A modified orthogonal fractional factorial design was employed to set up simulation cases to investigate the influence of parameters (computational domain size and mesh refinement) on results of interest (flow field and pressure field). The conclusions of this case study are as follows:

- Mesh construction has undue influence on flow field;
- Upstream length and mesh refinement significant for simulating the velocity field;
- Upstream length and cross-stream width important for simulating the pressure field;
- Upstream length significant for simulating vortex shedding frequency;
- Downstream length does not appear to have a significant influence of any of the flow parameters considered except downstream velocity and base pressure. However, in both cases, its influence is relatively minor.

Once the numerical errors associated with the parameters are minimized or within acceptable level, the simulation based on the optimal set of parameters can be validated with experimental/numerical results of the same problem. Note that this sensitivity information cannot be applied to simulations that use different simulation details.

In light of these conclusions, further research is required to investigate the shortcomings of the current model configuration. It is recommended that further research be carried out to identify the appropriate turbulence model for the bluff body aerodynamics, as well as the parameters associated with the 3D simulations, such as spanwise length and spanwise grid size.

ACKNOWLEDGEMENTS

The authors would like to thank Prof. Guest and Prof. Schafer in the Civil Engineering Department of Johns Hopkins University for facilitating the running of these simulations on their high performance Linux cluster and their continued support of this research. The authors are also indebted to Dr. James J. Filliben in National Institute of Standards and Technology for his valuable advice on the experimental design technique.

REFERENCES

Bearman, P. W., Trueman, D. M., (1972). An investigation of the flow around rectangular cylinders. Aeronautical Quarterly, 23, 229-237.

Bearman, P. W., Obasaju, E. D., (1982). An experimental study of pressure fluctuations on fixed and oscillating square-section cylinders. Journal of Fluid Mechanics, 119, 297-321.

Box, G. E. P., Hunter, W. G., Hunter, J. S., (1978). Statistics for experimenters: an introduction to design, data analysis and model building. New York, NY: John Wiley & Sons.

Dahl, S. M., (2014). Unsteady RANS simulation of flow around rectangular cylinders with different aspect ratios at high Reynolds number (Master Thesis). Department of Marine Technology, Norwegian University of Science and Technology.

Durao, D. F. G., Heitor, M. V., Pereira, J. C. F., (1988). Measurements of turbulent and periodic flows around a square cross-section cylinder. Experiments in Fluids, 6, 298-304.

Franke, R., Rodi, W., and Schönung, B., (1990). Numerical calculation of laminar vortex-shedding flow past cylinders. Journal of Wind Engineering and Industrial Aerodynamics, 35, 237-257.

Lee, B. E., (1974). The effect of turbulence on the surface pressure field of a square prism. Journal of Fluid Mechanics, 69(2), 263-282.

Lee, S., Bienkiewicz, B., (1998). Finite element implementation of large eddy simulation for separated flows. Journal of Wind Engineering and Industrial Aerodynamics, 77&78, 603-617.

Lyn, D. A., Rodi, W., (1993). The flapping shear layer formed by flow separation from the forward corner of a square cylinder. Journal of Fluid Mechanics, 267, 353-376.

Lyn, D. A., Einav, S., Rodi, W., Park, J. H., (1995). A laser-Doppler velocrimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder. Journal of Fluid Mechanics, 304, 285-319.

Nishimura, H., Tanijke, Y., 2000. Fluctuating wind forces on a stationary two-dim square prism. 16th Wind Engineering Symposium in Japan, 255-260.

Noda, H., Nakayama, A., 2003. Free-stream turbulence effects on the instantaneous pressure and forces on cylinders of rectangular cross section. Experiments in Fluids, 34 (2003), 332-344.

The OpenFOAM Foundation, 2016. "OpenFOAM - Free, open source software from the OpenFOAM Foundation", Retreived, 7th May 2016 (http://openfoam.org).

Oberkampf, W. L., Trucano, T. G., 2008. Verification and validation benchmarks. Nuclear Engineering and Design, 238, 716-743.

Roache, P. J., 1997. Quantification of Uncertainty in Computational Fluid Dynamics. Annual Reviews Fluid Mechanics, 29:123-60.

Shimada, K., Isihara, T., (2002). Application of a modified k-ε model to the prediction of aerodynamic characteristics of rectangular cross-section cylinders. Journal of Fluids and Structures, 16(4), 465-485.

Tamura, T. Yoshiyuki, O., (2003). LES analysis on aeroelastic instability of prisms in turbulent flow. Journal of Wind Engineering and Industrial Aerodynamics, 91, 1827-1846.

Tian, X., Ong, M. C., Yang, J., Myrhaug, D., (2012). Unsteady RANS simulations of flow around rectangular cylinders with different aspect ratios. Ocean Engineering, 53, 208-216.