Mitigation of Wind Induced Vibration in Substation Structures: A Fluid Structure Interaction Study on Curved Spoilers

Diaaeldin Mohamed, Ph.D., P.E.¹; and William J. Reisdorff, Jr., P.E.²

¹Valmont Utility, PO Box 358, 28800 Ida St., Valley, NE 68064, USA. E-mail: diaaeldin.mohamed@valmont.com
²Valmont Utility, PO Box 358, 28800 Ida St., Valley, NE 68064, USA. E-mail: william.reisdorff@valmont.com

ABSTRACT

As modern utility structures move toward taller and more flexible designs, the problems of wind effects on these structures have become increasingly apparent. While wind induced vibration is a common phenomenon that can lead to structural failure, it is one that is often overlooked because of the complexity of modeling the situation correctly. Vortex-induced vibration (VIV) is more complex than a mere resonant forcing problem. The complexity and uncertainty of the wind field and its interaction with structures that necessitate such an interdisciplinary approach, involving scientific fields such as meteorology, fluid dynamics, statistical theory of turbulence, structural dynamics, and probabilistic methods. Many structures used in substations (e.g. A-frames and static masts) can be prone to this problem. Based on some field reports of structural vibration, a study is performed to better understand the VIV phenomenon and find ways for prediction and mitigation. This paper presents a numerical analysis to predict and mitigate the VIV effects using ANSYS Workbench two-way 3D Fluid Structure Interaction (FSI). FSI is a coupling process between Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA). This paper concludes with some recommendations on how to mitigate VIV in substation structures using curved spoilers, in addition to some considerations that need to be taken into account during the design of these new structures and a retrofit plan for in-service structures as well.

Keywords
Utility, substation, wind-induced vibration, vortex shedding, ANSYS, mitigation, curved spoilers

INTRODUCTION

Vortex Shedding (also known as Instability-Induced Excitation (IIE)) is associated with flow instability and involves local flow oscillations. In general, vortex shedding phenomenon happens when wind with critical Reynolds number hits a bluff body (e.g. utility structure), causing alternating vortices to form at a certain frequency. This in turn causes the whole structure (or in some cases, members of the structure) to excite and produce a vibrational load. This load happens because the boundary layers shed alternatingly from the structure with a vortex shedding frequency. This frequency is related to the speed at which vortices are convicted and the separation distance between vortices. The change in momentum due to shed fluid vortices imparts a lift force in the cross-flow direction along the structure and this is why the structure vibrates dynamically under cyclic loading.

The extreme case of shedding frequency excitation occurs when the shedding frequency aligns with a natural frequency of the structure. This condition is known as “lock-in”. The mode is excited strongly with very little damping. The peak response at the resonance frequency jumps upward because of the loss of damping. In this scenario, structures may undergo a great number of stress cycles that lead to damage accumulation and may determine structural failure without exceeding the ultimate limit stress. Crack
initiation and crack growth phases happen in this scenario leading to fatigue and ultimate catastrophic failure of the structure. Tall slender utility structures with no wires or cables are highly susceptible to vibrations induced by vortex shedding.

Accurate analysis of a complex problem like vortex shedding is seldom possible by using only computational or experimental techniques and instead requires the joint application of different techniques. For instance, wind tunnel and full-scale experiments frequently provide the input for theoretical solutions, numerical simulations and code provisions; theoretical and numerical methods are often the bases for organizing, developing and processing experimental measurements. In general, powerful analytical and simulation tools are required when investigating complex wind induced phenomenon like vortex shedding.

Mitigating wind-induced vibration via experimental work is limited by the amount and data collected from wind tunnel test. For instance, detailed flow fields and structural features, such as velocity field data, surface pressure and structural stresses, are very difficult to measure with experiments. On the other hand, numerical models and simulations can decrease time and costs associated with experimental approaches.

CFD is a promising alternative to wind tunnel testing because it involves the solution of the Reynolds-Averaged Navier-Stokes (RANS) equations in the Large Eddy Simulation (LES) framework to simulate pressure fields around structures that convincingly reproduce the experimentally measured pressure distributions in both the mean and RMS, as well as replicating the aerodynamic forces and flow re-attachment features. The combination of RANS and LES equations results in a new turbulence model called Detached-Eddy Simulation (DES) which uses RANS equations for the near wall treatment and LES equations for the rest of the flow field. This research uses DES turbulence model to numerically simulate the VIV phenomenon. FEA and CFD solvers in ANSYS Workbench commercial suite are used in this work to perform a three-dimensional two-way numerical FSI modeling on a full-scale A-frame.

**Fluid-Structure Interaction (FSI) Problems**

FSI modeling is a method by which fluid (e.g. wind) and solid domains (e.g. structure) are coupled together to produce a single result that cannot be produced if each physical domain is evaluated individually [1-4]. Traditionally, the phenomena of fluid dynamics, heat transfer, solid mechanics, electro-mechanics, electromagnetics, vibrations, and chemistry have been evaluated separately [4-7]. However, with more powerful computers, commercially available software, and new techniques to add computational stabilities, engineers can now apply computational techniques to increasingly complex systems by linking multiple domains and analysis techniques together [1]. This increased complexity is evident both in the model size and in the ability to capture the full multi-physics environment.

FSI links CFD and FEA models together so that the results of each model impart forces on the other. CFD and FEA domains require separate meshes and numerical procedures to compute their respective solutions. In order for these domains to interact, an interface is needed that allows data transfer between the domains. The transferring of data (e.g. displacement, force, pressure, and temperature) back and forth between the fluid and solid domains is also an important feature of FSI modeling.

**Data Transfer Methods**

Data transfer between the CFD and FEA domains requires defining the frequency and direction of shared information as it is passed. The frequency by which information is passed is defined by the coupling type: monolithic coupling, strong coupling, or weak coupling [2, 8-10], while the type and direction of the information passed can be one-way or two-way.

Monolithic coupling involves solving both the fluid and solid system of equations simultaneously as a single system/matrix of equations. Generally, monolithic data transfer utilizes custom computational codes and requires extremely large and powerful supercomputers compared to other coupling methods [5]. In
contrast, both strong and weak couplings pass data between the fluid and solid models in an attempt to solve the two systems of equations separately, but with shared boundary conditions [10-16].

The difference between strong and weak coupling methods is when and how often data is passed from one model to another with respect to each time step. In strong coupling, each domain is evaluated once and then the data is exchanged between the models. Then the same time step is re-evaluated using the results from the other domain as updated boundary conditions. This process of exchanging data between the domains is repeated until a converged solution is reached in both domains, then the next time step is taken and the data exchange process repeats. In weak coupling, data is exchanged a maximum of one time between domains before the next time step is taken thus no check is performed to ensure a converged data transfer has been reached. In addition to not checking for a converged data transfer, the data may be transferred less frequently, which leads to very weak coupling and eventually one-way coupling.

One-way coupling only passes data in one direction, meaning the model either passes data from the fluid to the solid or from the solid to the fluid. One-way coupling is a useful tool to reduce the computational time required to evaluate a model [2-4]. However, this coupling can only be utilized when the results of one model will have insignificant effects on the other model, which is not the case in studying a phenomenon like vortex shedding, fig. 1-a. Two-way coupling is required when data is passed in both directions between the fluid and solid models [2-4]. This type of coupling is necessary for models with large deformations like wind induced vibration because the results of one model will significantly alter the boundary conditions and outcome of the other, fig. 1-b.

![Diagram of one-way and two-way coupling](image)

(a) One-way coupling  (b) Two-way coupling

**Figure 1.** FSI data transfer of information in vortex shedding simulation

Regardless of the coupling method, when CFD and FEA models are coupled together, two challenges are introduced. The first challenge is coupling the two independent mesh domains together while still accounting for the differences in mesh formulation and motion. This challenge arises from the fundamental differences between the Lagrangian mesh that is utilized in FEA models where the finite element mesh is fixed to the mass and moves in space as a function of the mass motion [8], and the Eulerian mesh that is utilized in CFD models where the finite element mesh is fixed in time and space with the mass passing through the mesh [1, 9]. This means that the Lagrangian mesh is able to deform and move positions as a function of the fluid-domain inputs; however, the solid domain displacement of the Lagrangian mesh cannot be directly applied to the fixed fluid domain. The second challenge is to transfer data between domains in
a manner that mitigates instabilities, fluctuations, and non-physical phenomena at the domain interfaces. These instabilities arise from the mass effect, data transfer methods, and magnification of instabilities or shock waves at the interface.

In this research, the FSI modeling is performed using FEA (ANSYS Mechanical Enterprise), CFD (ANSYS CFD Enterprise), and Fatigue (ANSYS nCode DesignLife) modules in ANSYS Workbench 2019 R2. The pre-processor used for generating the geometry and mesh is ANSYS DesignModeler. Figure 2 shows a flow chart of a two-way 3D FSI modeling in ANSYS Workbench.

![Flow Chart of Two-Way 3D FSI Modeling in ANSYS Workbench](image)

**Figure 2.** Two-way 3D FSI modeling in ANSYS WORKBENCH

(1- Natural Frequency Analysis (MODAL), 2- FEA Solver (Transient Structure), 3- CFD Solver (FLUENT), 4- FSI System Coupling, 5- FSI Results, 6- Fatigue Solver (Strain Analysis), 7- Fatigue Solver (Stress Analysis), and 8- Fatigue Solver (Vibration Analysis))

**Complexity of Two-Way 3D FSI Modeling**

All FSI models introduce challenges that can result in an inaccurate solution and computational divergence. In particular, repeatedly using the output of a computational model as the input to another computational model can result in compounding errors from repeatedly using the same slightly incorrect values. Consequently, the more times data is passed, the larger the compounding error. FSI models also experience convergence issues caused by the mass effect, residual convergence between each domain, possible ramping of data between each interface, methods by which data is passed between interfaces, frequency by which data is passed between interfaces, and magnification of any instabilities at the interface. Furthermore, FSI modeling also experiences the same instabilities and convergence challenges as individual CFD and FEA models.

**MOTIVATION**

Commercial software packages are currently available that allow engineers to produce complex FSI models. Many publications are available that utilize these packages on an application basis to evaluate the design of existing systems. However, to the authors’ knowledge, a complete qualitative and quantitative validation of these commercially available software packages does not exist. This research sets out to provide a complete qualitative and quantitative validation and then test the performance accuracy of the models by evaluating design alternatives both computationally and experimentally. Validation is paramount to ensure the FSI model accurately represents the physical system. Therefore, an experiment to monitor a full-scale
A-frame in the field was conducted to validate the performance of the curved spoilers and is published separately in a different paper.

**PROBLEM DESCRIPTION**

This research investigates the curved spoilers as a vortex shedding mitigation approach in substation structures. A three-phase dead-end A-frame located in the Midwest was reported with cyclic vibration in its four legs, fig. 3-a. The structure has a height of 175 ft and designed for 3 kips of wire tension and 24 kips of cable tension on each phase with optional switch and short beams, fig. 3-b. Only the short beams were added after the dynamic vibration started to temporary minimize the deflection magnitude in the legs while a final solution is being developed in Valmont Utility. The A-frame members are manufactured from ASTM A572 Grade 65 save for the base plates are manufactured from ASTM A572 Grade 50.

![Figure 3. A schematic of the A-frame](image)

(a) Front and side views of the A-frame  
(b) Tension in static wires and conductor cables

**ANSYS Model**

Unlike designing a mitigation approach for a new structure, adding spoilers/strakes to in-service structures is very tricky because it results in increasing the wind pressure because the spoilers/strakes would increase the wind projection area. Accordingly, several studies are performed to determine the final shape of the spoilers and numerous 2D and 3D CFD models are tested initially to judge which geometry would disrupt the creation of vortices in the wake region and reduces the lift forces to a percentage that would eliminate the dynamic vibration in the legs and extend the fatigue life of the whole structure but in the meantime the spoiler geometry would not increase the drag forces to a percentage that would reduce the static design safety factor per ASCE 48-11 below 1.0. Figure 4 shows the final design of the curved spoilers. Each spoiler has a width of 12 inch and a height of 48 inch. The width is bent twice at 45 and 90 degrees, resulting in approximately three 4 inch long edges. Each spoiler is manufactured from A 36 structural steel and has a weight of 30 lbs. Spoilers were added in a helical pattern per ASME STS-1 (steel chimney standard), fig. 5. Sixty spoilers were installed per leg on every other flat. Since the legs are 12-sided, spoilers were added as six columns of ten spoilers per column. Each spoiler is attached by two Rivnuts 36 inches apart.
**Figure 4.** A schematic of the curved spoilers

**Figure 5.** A schematic of curved spoilers attached in a helical pattern

**CFD Fluid Domain**

The CFD computational model containing the fluid domain utilizes the three-dimensional Navier–Stokes equations in conjunction with the continuity, and energy equations. Each of CFD equations is defined for atmospheric air conditions at a control volume prescribed by the fluid mesh, resulting in one set of equations being evaluated over the entire domain. Detached Eddy Simulation (DES) turbulence model is used along with a second order implicit pressure-velocity coupling to solve the CFD domain. Time step is set to 0.0001 second and maximum number of iterations per time step is set to 100. The CFD model is evaluated using standard relaxation for pressure, density, body forces, momentum, turbulence kinetic energy, turbulent dissipation rate, and turbulent viscosity of 0.3, 1.0, 1.0, 0.7, 0.8, 0.8, and 1.0 respectively, until all scaled convergence values were below 0.001.
The fluid domain was initially evaluated independently of the solid domain and without dynamic meshing. This uncoupled CFD model allowed for greater understanding of the mesh cell size sensitivity, convergence criteria as a function of flow rate, and required convergence time as a function of time step size and number of iterations. The information gathered from evaluating just the CFD model without dynamic meshing provided valuable insight into what time step size and flow rate allowed the optimal balance between a reliably stable fluid solution and overall computational time required to evaluate the model.

Additionally, a mesh density investigation of the CFD domain was performed when the fluid domain was uncoupled in order to explore the proper mesh density and assess which portions of the model required a finer mesh and which regions could tolerate a coarser mesh. Because a large fluid domain was present, it was not desirable to have a uniformly fine mesh over the entire domain. The mesh density investigation indicated the size and regions where course and fine mesh required implementation. Therefore, eighteen mesh zones were used to add flexibility in using coarse and fine mesh elements and to limit the number of tetrahedron elements, see fig. 6. The size of the three-dimensional mesh is approximately 15 million hexahedron and tetrahedron elements, with an inflation of ten to fifteen layers around the A-frame outer walls but this number varies as the structure deforms and dynamic meshing occurs, fig. 7.

![Figure 6. CFD domain](image)
Figure 7. Hexahedron and tetrahedron elements in the CFD domain

FEA Solid Domain

The FEA computational model utilizes the three-dimensional strain displacement, nodal displacement, and stress equations, to solve for the deformation, stress, strain, and forces across each node in the solid domain. The solid mesh contains approximately 3 million hexahedron and tetrahedron elements, with an inflation of five to ten layers around the A-frame outer walls.

The solid domain was initially evaluated uncoupled from the fluid domain using NESC Heavy Wind and Ice load case applied to static wire and conductor arms (SWL, SWR, C1, C2, and C3), fig. 8. This uncoupled model was used to understand the stability of the structural steel and connections between the A-frame and the spoilers in order to determine proper time step size, and perform a mesh density study. The final coupled domain is studied unstrung (no wires or cables attached to the A-frame to represent the worst case scenario with no damping applied to the structures whatsoever).

(a) Isometric view  (b) Side view  (c) Front view
Interface between Fluid and Solid Domains

Upon completion of modeling CFD and FEA models separately, the models were coupled together and dynamic meshing was enabled in the CFD model. Table 1 lists the boundary conditions used in the CFD and FEA models. User defined functions (UDFs) are used in the CFD model to define variable wind speed from zero to 35 mph.

Table 1. Boundary conditions (BCs) for the FSI model

<table>
<thead>
<tr>
<th>Domain</th>
<th>BC Name</th>
<th>BC Type</th>
<th>Settings</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fluid</td>
<td>Inlet</td>
<td>Velocity Inlet</td>
<td>Variable Wind Speed (0-35 mph)</td>
</tr>
<tr>
<td>Fluid</td>
<td>Outlet</td>
<td>Pressure Outlet</td>
<td>Atmospheric Pressure</td>
</tr>
<tr>
<td>Fluid</td>
<td>A-frame</td>
<td>Wall</td>
<td>No slip and coupled with dynamic meshing</td>
</tr>
<tr>
<td>Solid</td>
<td>A-frame</td>
<td>Fluid structure Interaction</td>
<td>Six degrees of freedom (three in translation and three in rotation)</td>
</tr>
<tr>
<td>Solid</td>
<td>Base Plates</td>
<td>Fixed Support</td>
<td>Fixed in six degrees of freedom (three in translation and three in rotation)</td>
</tr>
</tbody>
</table>

All computational studies are performed on a very expensive desktop workstation (DELL Precision 7920 Tower that is valued at about 55k USD), table 2. ANSYS High Performance Computing (HPC) license with 32 cores is used to utilize most of the workstation computational capabilities. MATLAB is used to manage the optimization and design of experiment for the CFD, FEA, and FSI models.

Table 2. Hardware of Dell Precision 7920 Tower

<table>
<thead>
<tr>
<th>Processor</th>
<th>Dual Intel Xeon Platinum 8280M 2.7GHz, 4.0GHz Turbo, 28C, 10.4GT/s 3UPI, 39MB Cache, HT (205W) 2.0T</th>
</tr>
</thead>
<tbody>
<tr>
<td>Memory</td>
<td>1TB 16x64GB DDR4 2666MHz LRDIMM ECC</td>
</tr>
<tr>
<td>Video Card</td>
<td>Triple NVLink NVIDIA Quadro GV100, 32GB, 4 DP</td>
</tr>
<tr>
<td>Hard Drive</td>
<td>4x 3.5&quot; 4TB 5400rpm SATA Hard Drive</td>
</tr>
</tbody>
</table>
RESULTS AND DISCUSSION

MODAL Analysis

A natural frequency study is performed to determine the excitation modes and the corresponding critical wind speeds for the original and spoilers’ models, table 3. Spoilers’ weight was not comparable to the weight of the original A-frame, therefore, the excitation modes remain the same. The main idea behind the spoilers is to disrupt the vortices created by the wind in the wake region rather than shifting the original natural frequency of the structure. This is a more favorable approach especially for mitigating VIV phenomenon in in-service structures or when changing the structure static design criteria is not an option.

<table>
<thead>
<tr>
<th>Excitation Mode</th>
<th>Frequency (Hz) [Original Model]</th>
<th>Frequency (Hz) [Spoilers Model]</th>
<th>Critical Wind Speed Range (mph)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.4</td>
<td>0.4</td>
<td>8 - 15</td>
</tr>
<tr>
<td>2</td>
<td>1.6</td>
<td>1.6</td>
<td>13 - 22</td>
</tr>
<tr>
<td>3</td>
<td>1.8</td>
<td>1.8</td>
<td>16 - 28</td>
</tr>
<tr>
<td>4</td>
<td>2.3</td>
<td>2.3</td>
<td>21 - 33</td>
</tr>
</tbody>
</table>

FSI Model

Two full-scale A-frame models (with and without curved spoilers) are studied using two-way 3D FSI modeling for variable wind speeds up to 35 mph in longitudinal and transverse directions to examine the effectiveness of the curved spoilers in mitigating the vortex induced vibration in the legs during all critical wind speeds determined by the MODAL analysis. Since the structure is modeled unstrung to investigate the worst case scenario, the results are symmetrical. Only results for the left side (SWL, C1, C2, LF, and LB) of the A-frame are presented below.

Longitudinal Wind

Even though curved spoilers are installed to mitigate the vortex shedding vibration in the legs only, they also helped reduce the deflection in the static wire and conductor arms and hence reducing the stress in the arms weld area. Spoilers added extra stiffness to the lower section by decreasing the dynamic vibration and overall deflection magnitude in the legs. As a result, transverse and vertical deflections (dynamic vibration due to wind lift forces) in SWL decreased by 40 and 17%, respectively, fig. 9-a. Transverse deformation in C1 and C2 decreased by 78, and 77%, respectively, figs. 9-b and 9-c, which would indicate less cyclic vibration for the arms (and the whole A-frame) in the cross-flow direction. In terms of longitudinal deformations (wind drag forces) in SWL, C1, and C2, spoilers added to the legs did not change the value.
Spoilers increased the drag forces on the legs due to increasing the projected surface area to the wind. Longitudinal deflection in LF and LB increased by 3.5%, fig. 10. Transverse deflection in LF and LB decreased by 80%, fig. 11.

Negligible change in equivalent stress is noticed in SWL, C1, and C2, fig. 12, because spoilers are added to the legs only to mitigate the dynamic vibration (however, spoilers can be added in a different shape/geometry to the static wire arms to mitigate wind induced vibration in them as well). Figure 13 shows how spoilers successfully convert dynamic stress (low stress – high cycles fatigue type) to a much more steady and static stress that ultimately can be compared against A572 grade 65 yield strength (65 ksi).
Transverse Wind

Longitudinal deflection in SWL, C1, C2 (due to wind lift forces) decreased by 71%, 14% and 15%, fig. 14. Transverse deflection (due to wind drag forces) in C1 and C2 increased by 5%, fig. 14-b and 14-c.

Transverse deflection in LF and LB increased by 5% due to increase of projection surface area in the legs, fig. 15. Longitudinal deflection in LF and LB decreased by 70% in the cross-flow direction, fig. 16.
Figure 15. Deformation in LF and LB due to drag forces

Figure 16. Deformation in LF and LB due to lift forces

Negligible change in equivalent stress is noticed in C1, and C2, while a maximum of 60% decrease in stress is observed in SWL at 35 mph wind, fig. 17. Figure 18 shows how spoilers successfully convert dynamic stress (low stress – high cycles fatigue type) to a much more steady and static stress that ultimately can be compared against A572 grade 65 yield strength (65 ksi).

Figure 17. Equivalent Stress in SWL, C1, and C1

Figure 18. Equivalent Stress in LF and LB

Effect of Curved Spoilers on Vortex Shedding Frequency

Although, only 10% change in the frequency response is recommended by many scientific articles to prevent ‘lock-in’ caused by wind induced vibration and extend the fatigue life of a structure, the curved spoilers resulted in a 43% change in the legs frequency response, fig. 19.
CONCLUSIONS

- Vortex shedding is a dynamic phenomenon and to get realistic structural responses to this phenomenon, it must be simulated dynamically. The dynamic structural response is the worst case scenario because of the realistic account for inertial induced loads. Estimating the wind-induced response of structures is usually carried out through theoretical formulations, numerical algorithms, wind tunnel tests, full-scale experiments and code provisions. Although an immense computational and experimental effort has been made during the last six decades to improve the analytical vortex shedding prediction models, the design standards are still lacking in presenting concise and easy to use analytical methodologies. Hence, such design procedures require the induced forces to be applied statically.

- While vortex shedding is a common phenomenon that can lead to structural failure, it is one that is often overlooked because of the complexity of modeling the situation correctly. Using FSI modeling, vibrational problems can be easily identified and hypotheses can be tested. The simulation methods described in this paper lay the foundation for how future FSI models should be constructed, evaluated, and validated.

- Producing an accurate FSI model requires far more than the full multi-physics and final computational models alone. Exploring and understanding the CFD and FEA models independently is paramount to understanding possible sources for instabilities, evaluating the models in the most efficient time possible, and validating the mesh, boundary conditions, and material properties. For this reason each model should be evaluated independently before evaluating the coupled model.

- The computational time required to evaluate an FSI model can be greatly reduced by: 1) selecting an appropriate time step, 2) only re-meshing regions with low quality elements, and 3) maintaining a non-uniform mesh size with course and fine regions.
Although design changes can be made easily for new structures before any real problem arises, it is considered a very difficult task as a retrofit plan; that is why numerous 2D and 3D CFD studies are conducted in this research to find the optimum spoiler shape that would eliminate the wind induced vibration in the legs to maintain the structure’s fitness for service even with a slight increase in the drag wind pressure.

When the A-frame experienced longitudinal wind, spoilers increased the deformation due to drag forces on the legs by only 3.5%, while decreasing the deformation due to lift forces by 80% in the cross-flow direction.

When the A-frame experienced transverse wind, spoilers increased the deformation due to drag forces on the legs by only 5%, while decreasing the deformation due to lift forces by 70% in the cross-flow direction.

Curved spoilers resulted in a 43% change in the legs frequency response exceeding the minimum 10% recommended target.

REFERENCES