Numerical Simulation of the divergence of a Wind Turbine Airfoil

Drishtysingh Ramdenee, Ion-Sorin Minea, Thierry Tardif D'Hamonville, Adrian Ilinca
Wind Energy Research Laboratory, Université Du Québec à Rimouski
300 allée des Ursulines, Rimouski, QC, G5L 3A1, Canada
Email: dreutch@hotmail.com

ABSTRACT

The recent development of large wind turbines poses new challenges with regard to understanding the mechanisms surrounding unsteady flow-structure interaction. The larger and more flexible blades imply risks from an aeroelastic point of view and urge the need to properly understand and model these phenomena. Due to limited experimental data available in this field, Computational Fluid dynamics (CFD) techniques provide an invaluable alternative to identify and model aerodynamic and aeroelastic phenomena around the wind blades. The study is part of the coupling between aerodynamic and elastic models of the commercial code - CFX with ANSYS, respectively. In this paper we are modeling the aeroelastic divergence. This article presents the second part of the studies aiming at modeling divergence. In this article the results of the divergence modeling using ANSYS-CFX will be presented and compared with results from "Jennifer Heeg" [3]. Several references will be made to the first part of this study: "D.Ramdenee, Ion Sorin Minea, T. D'Hamonville, A. Ilinca" [4]. The study is realized on the NACA0012 airfoil for which experimental data are available in literature. The ANSYS workbench is used for the fluid structure interaction to simulate the divergence phenomenon which is a structural response imposed by aerodynamic loads due to transient fluid flow.

1. INTRODUCTION

In an attempt to increase power production and reduce material consumption, wind turbines are becoming increasingly gigantic and, yet, paradoxically, thinner and more flexible. The wind turbines' blades are, thus, more and more prone to deflections and vibrations due to forces generated by the wind. These phenomena are known as aeroelastic phenomena and are subject of many investigations. "T. Tardif d'Hamonville, A. Ilinca" [1, 2]. The modeling of these phenomena is done by the coupling of aerodynamic models and elastic models. This article presents a study realized on the modeling of the divergence phenomenon on wind turbines' blades. This study is realized on the NACA0012 airfoil and using the k-ω SST turbulent model in ANSYS-CFX computational fluid dynamics software. The importance of such modeling and the scope and pertinence of such an aeroelastic project is presented in depth in “D.Ramdenee, Ion Sorin Minea T. Tardif d’Hamonville, A. Ilinca” [7].

2 DEVELOPMENT OF THE ARTICLE

This article presents a study conducted on divergence by “Jennifer Heeg” [3] and aims at reproducing a model on ANSYS that is as close as possible to the experiment model and try to compare results obtained in [NASA] with results generated using ANSYS - CFX. The turbulence and transitional models used in this study as well as the analytical development of the equations used in this study are explicitly in “D.Ramdenee, Ion Sorin Minea T. Tardif d’Hamonville, A. Ilinca” [7].

3 EXPERIMENT TO BE COMPARED WITH

3.1 Overview of Experiment

An aeroelastic experiment was conducted in the Duke University Engineering wind tunnel facility. The goals of this test were to validate the analytical calculations of noncritical mode characteristics and to explicitly examine the aerodynamic mode divergence phenomenon. To these ends, the simplest applicable model that could be devised was designed, fabricated and tested.

3.2 Configuration Description

The divergence assessment testbed (*dat*) wind tunnel model consists of a typical section airfoil with a flexible mount system providing a single degree of freedom structural dynamic mode. The only structural dynamic mode of this model is torsional rotation, or angle of attack.

The airfoil section is a NACA 0012 with an 8-inch chord and a span of 21 inches. The ratio of the trailing edge mass to the total mass is 0.01. This spans the entire test section from the floor to ceiling. The structural dynamic parameters for this model are illustrated in table 1:

K_α (N·m/rad)	ω_α (rads/sec)	**f_α** (Hz)	ζ
5.8262	49.5	7.88	0.053

Table 1: Excerpt from Table 5 in “Jennifer Heeg” [3]: Structural dynamic parameters associated with wind tunnel model configurations

Table 2 lists the analytical calculations for divergence conditions for the considered model presented in [Heeg]:

Velocity			Dynamic Pressure	
(in/sec)	(mph)	(m/s)	(psf)	(N/m^2)
754	42.8	19.15	4.6	222

Table 2: Excerpt from Table 6 in “Jennifer Heeg” [3]: Analytical calculation of divergence conditions

However, some parameters were unavailable in “Jennifer Heeg” [3] such that an iterative design process was used to build the model used in ANSYS. Using parameters specified in “Jennifer

Heeg” [3], a preliminary model was built and its natural frequencies verified using ANSYS. The model was successively modified until a model as close as possible to the model in the experiment considered in “Jennifer Heeg” [3] was obtained.

The aims of the studies conducted in “Jennifer Heeg” [3] were to: 1) to find the divergence dynamic pressure; 2) to examine the modal characteristics of non-critical modes, both subcritically and at the divergence condition; 3) to examine the eigenvector behavior. “Jennifer Heeg” [3] proposes several interesting results among which the following which is a graphic showing the variation of the angle of attack with time. The aim of our simulations is to be able to simulate the same using ANSYS-CFX.

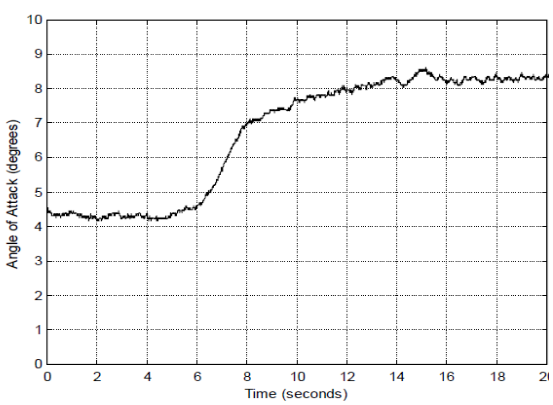

Figure 1: Divergence of wind tunnel model configuration #2

The test was conducted by setting as close as possible to zero the rigid angle of attack, α_0, for a zero airspeed. The divergence dynamic pressure was determined by gradually increasing the velocity and measuring the system response until it became unstable. The dynamic pressure was being slowly increased until the angle of attack increased dramatically and suddenly. This was declared as the divergence dynamic pressure, 5.1 psf (244 N/m2). The time history shows that the model oscillates about a new angle of attack position, which is not at the hard stop of the spring. It is speculated that the airfoil has reached an angle of attack where flow has separated and stall has occurred “Jennifer Heeg” [3].

4.1 The ANSYS-CFX model

The model of the experiment was simulated at a reduced scale, in order to reduce the calculation time by reducing the dimensions of the fluid domain. The span of the airfoil was reduced 262.5 times, from 21 inches to 0.08 inches or 2.032 mm, while the chord of the airfoil was maintained at 8 inch or 203.2 mm. We used a cylinder to simulate the torsion spring used in the configuration of the experiment that was detailed in the previous section. The constant of the original spring is $K_\alpha = 5.8262$ N·m/rad and since we used a reduced model, with an span 262.5 times smaller than the original, the dimensions and properties of the cylinder are such that:

$$K_{\alpha r} = \frac{5.8262}{262.5}\ \mathrm{N \cdot m/rad} = 0.022195\ \mathrm{N \cdot m/\ rad}$$

The mass of the considered configuration of the original model is 2.2864 kg, and the mass of our model is 262.5 times smaller, that is 0.00871 kg.

The moment of inertia is such that our model has the same fundamental frequency as the original that is 7.88 Hz.

Figures 2 and 3 illustrate the model built on ANSYS. Figure 2 shows exclusively the geometry while figure 3 illustrates also the meshes on the model.

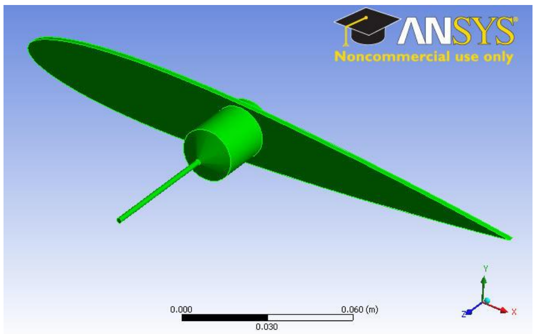

Figure 2: ANSYS built geometry for the NACA0012 profile

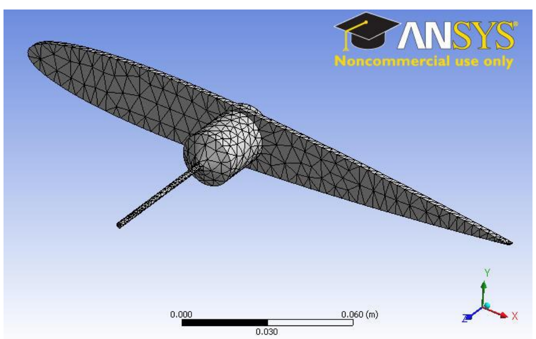

Figure 3: ANSYS built model with meshes

4.2 Fluid Domain Model

The fluid model was built based on works conducted by “T. Tardif d’Hamonville, A. Ilinca” [1, 2]

In this study we work with a domain defined by a semi disc with radius I_1*c around the profile and two rectangles in the wake of length I_2*c as illustrated in figure 4.

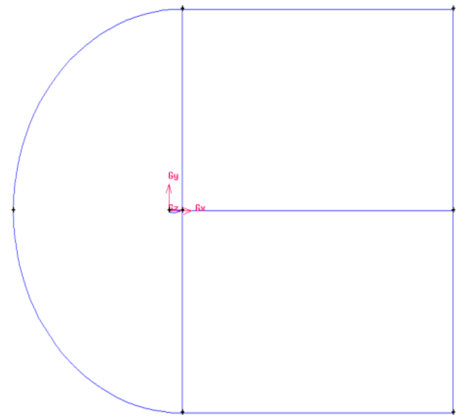

Figure 4: Domain Illustration

In this case, the values of these lengths are given in table 2 below:

Parameter	I_1	I_2	c
Length (m)	9	15	0. 2032

Table 2: Characteristic lengths of the fluid model

4.3 Characteristics of the fluid domain mesh

The characteristics of the elements along the profile are presented below:

The size of the elements along the profile, a1 (m) is set between 0.0005 and 0.001.

The growth rate of the elements towards the external limits of the domain: f2 = 1.2

Size of the elements at the external limits of the domain: a7 (m) = 0.15

Height of the first element of the boundary layer: a3 (m) = 0.00002

Number of divisions in the boundary layer: n3 = 9

Expansion factor of the elements of the boundary layer: f1 = 1.16

5. RESULTS

"D.Ramdenee, Ion Sorin Minea, T. D'Hamonville, A. Ilinca" [4] derived the analytical mathematical equation to calculate the divergence velocity.

The expression was the following:

$$U_D = \sqrt{\frac{1}{C^{\theta\theta}\frac{\partial C_l}{\partial \alpha}e\frac{1}{2}\rho S}} \quad (1)$$

In order to calculate the theoretical value of the divergence velocity, certain parameters need to be found first. These are $C^{\theta\theta}$, which is specific to the modeled spring, S being inherent to the profile, e, which depends both on the profile (elastic axis) and on the aerodynamic model, ρ, which is dependent upon the used fluid and $\frac{\partial C_l}{\partial \alpha}$ which depends both on the shape of the profile but also on the used turbulent model as explained in "D. Ramdenee, A. Ilinca" [5].

We note that "T. Tardif d'Hamonville, A. Ilinca" [6] as divergence velocity is approached, the elastic twist angle will increase in a very significant way and tend to infinity. However, softwares are finite and cannot model infinite parameters. We will, therefore, formulate the value of the analytical elastic twist angle in order to compare it with the value found by the coupling. In the case wherein the elastic twist angle introduces no further aerodynamic solicitations, by introducing $\alpha = \alpha_r$, and resolving for the elastic twist angle, we have:

$$\theta_r = C^{\theta\theta}T = C^{\theta\theta}(\frac{\partial C_l}{\partial \alpha} e\,\alpha_r + C_m c)qS \quad (2)$$

Dividing equation (5) fom [Numerical Simualtion_part1] by equation (2):

$$\theta = \frac{\theta_r}{1-\frac{\partial C_l}{\partial \alpha}C^{\theta\theta}eqS} \quad (3)$$

This leads to:

$$\theta = \frac{\theta_r}{1-\frac{q}{q_D}} = \frac{\theta_r}{1-(\frac{U}{U_D})^2} \quad (4)$$

Hence, we can note that the theoretical elastic twist angle depends on the divergence speed and the elastic twist angle calculated whilst considering that it triggers no supplementary aerodynamic solicitation. In order to calculate the latter, we will solve for the moment applied on the profile at the elastic axis (T) during trials in steady mode. These trials are conducted using "T. Tardif d'Hamonville, A. Ilinca" [2], the k-w SST intermittency transitional turbulence model with a 0.94 intermittency value.

In order to obtain the flexibility coefficient of the rotational spring, $C^{\theta\theta}$, used in the NASA experiments for the first two configurations we used a cylinder as a torsion spring. The constant of the spring used in the experiment is K_α = 5.8262 N·m/rad and since we used a reduced model, with an span 262.5

times smaller than the original, the dimensions and properties of the cylinder are such that:

$K_{\alpha r} = \frac{5.8262}{262.5}\ N \cdot m/rad$ = 0.022195 *N· m/ rad*

And the flexibility coefficient is:

$$C^{\theta\theta} = \frac{1}{K_{\alpha r}} = 45.0552\ rad/N \cdot m$$

The slope of the lift profile $\frac{\partial\ C_l}{\partial \alpha}$, can be calculated for an angle, $\alpha = 5^0$ in the following way:

$$\frac{\partial\ C_l}{\partial \alpha} = \frac{C_{l,\alpha>5^0} - C_{l,\alpha<5^0}}{\alpha>5^0 - \alpha<5^0} \qquad (5)$$

We have calculated the lift coefficient at the angle of 4.0^0 and 6.0^0 such that:

$$C_{l,\alpha=4.0^0} = 0.475$$

And

$$C_{l,\alpha=6.0^0} = 0.703$$

Such that:

$$\frac{\partial\ C_l}{\partial \alpha} = \frac{0.703 - 0.475}{6.0 - 4.0} = 0.114\ deg^{-1} = 6.532\ rad^{-1}$$

The distance, e, between the elastic axis and the aerodynamic centre for the model we considered in [numerical Simulation_part1] is 0.375·b, where b = 0.

The rigid area is calculated to be S, being the product of the chord and the span:

S = 0.2032·0.5334 = 0.0004129024 m^2.

Hence the divergence velocity is calculated as:

$$U_D = \sqrt{\frac{1}{C^{\theta\theta}\frac{\partial\ C_l}{\partial \alpha}\frac{\rho}{2}eS}} = 18.78 \text{ m/s} \qquad (6)$$

The theoretical divergence speed given in Table 2 of the article presenting the NASA experiment is 19.15 m/s. This slightly difference is due to the value of slope of the lift profile $\frac{\partial\ C_l}{\partial \alpha}$ taken into consideration, which in the NASA work was 2π, or 6.283 rad^{-1}, whereas we used a value of 6.532 rad-1

Furthermore, a difference between our calculated speed and that presented in "Jennifer Heeg" [3] might also be explained by the size of the used tunnel and the possible wall turbulence interaction that might have occurred.

6. PRELIMINARY RESULTS

Simulations are currently being run to model the divergence phenomenon. The future of the project is discussed in part 7 of this article.

However, preliminary results of the simulations aiming at modeling divergence are presented in this section. Using the model, domain and mesh parameters detailed in the previous sections of this article, divergence modeling using ANSYS – CFX structure illustrated in "D.Ramdenee, Ion Sorin Minea, T. D'Hamonville, A. Ilinca" [4] has been performed. The profile used by "Jennifer Heeg" [3] was fixed and exempted from all rotational degrees of liberty and subjected to a constant flow of velocity 15 ms^{-1}. Suddenly, the fixing is removed and the constant flow can be then compared to a shock wave on the profile. The profile then oscillates with damped amplitude due to the aerodynamic damping imposed. Figure 5 illustrates the response portrayed by ANSYS- CFX software. We can extract the amplitude of oscillation and a frequency of oscillation of around 8 Hz which is close to the 7.9 Hz frequency presented in “Jennifer Heeg” [3]

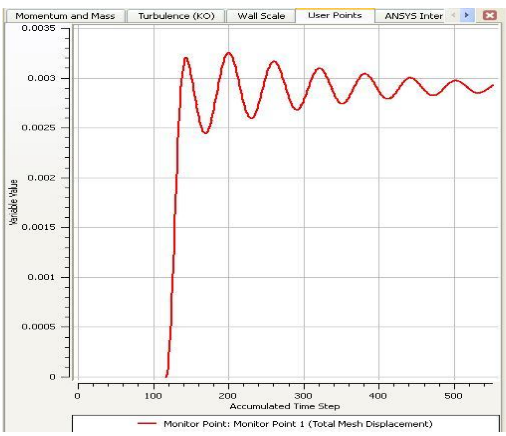

Figure 5: Oscillatory response to sudden subject to a constant flow of 15ms-1

For this speed, we have managed to compare the frequencies presented in “Jennifer Heeg” [3]. However, experiments have, also, been conducted for velocities of 24 m$s^{-1}$and 25 ms^{-1}. These results have not yet been calibrated and will not be discussed but only presented. The analysis of these results and comparison with other experimentation, analytical values or simulations will be done in the future.

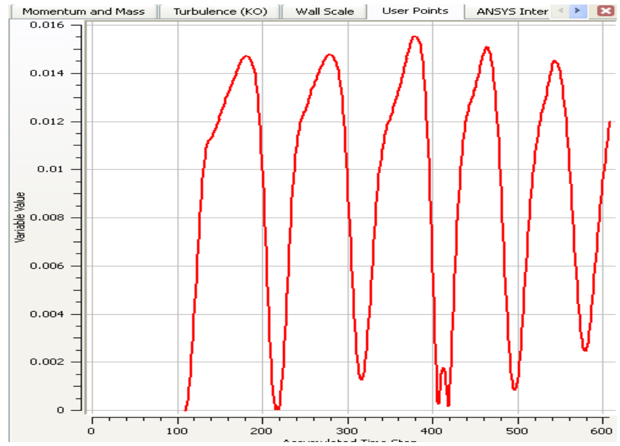

Figure 6: Oscillatory response to sudden subject to a constant flow of 24ms-1

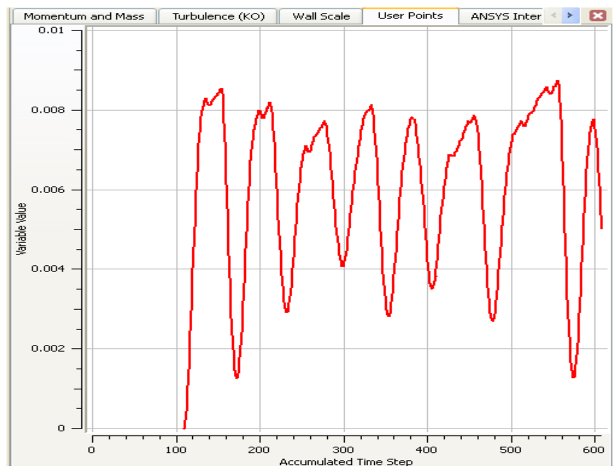

Figure 7: Oscillatory response to sudden subject to a constant flow of 25ms-1

7. ONGOING PROJET

This aeroelastic project is an ongoing and currently simulations are being done on the presented ANSYS model to simulate the divergence phenomenon. The ANSYS-CFX set up and the turbulence models presented in “D.Ramdenee, Ion Sorin Minea, T. D’Hamonville, A. Ilinca [4] are being used. The model is subjected to a flow of constantly increasing velocity. The phenomenon is highly complex and requires a constant exchange of information from the structural module of the workbench to the fluid module (CFX) and back. Furthermore, it has been noticed that in order to correctly model the phenomenon very small time steps are required. These two reasons make the modeling of the divergence phenomenon very long and tedious.

We expect to extract more pertinent and precise results from the above mentioned simulations in a few weeks and will be compared with the results presented in "Jennifer Heeg" [3].

8. CONCLUSION

The first part of this study "D.Ramdenee, Ion Sorin Minea, T. D'Hamonville, A. Ilinca" [4] and this article combine the different efforts to model divergence. The first article provides an explanation of the phenomenon, derives the analytical equations to calculate the divergence velocity and the eigen values related to the phenomenon. Furthermore, the first part of the article makes a broad literature review of the phenomenon and provides background information about the structure of ANSYS-CFX used for our particular application and a summary of the calibration done on the tools to determine the relative performance of the different turbulence and transitional models offered in the software. This article, which is the second part of the study, has focused on the creation of an ANSYS model which relates as closely as possible to the one used in "Jennifer Heeg" [3] and the validation of the analytical development with the results presented in "Jennifer Heeg" [3]. Furthermore, this article has presented the results from "Jennifer Heeg" [3] with which, we wish to compare the oncoming results from simulations conducted on ANSYS- CFX.

REFERENCES

[1] T. Tardif d'Hamonville, A. Ilinca. Modélisation de l'écoulement d'air autour d'un proil de pale d'éolienne,Phase 2 : Modèles Turbulents.
Rapport Technique , UQAR/LREE-06, Janvier 2009

[2] T. Tardif d'Hamonville, A. Ilinca. Modélisation de l'écoulement d'air autour d'un proil de pale d'éolienne,Phase 1: Domaine étude et Maillage.
Rapport Technique , UQAR/LREE-05, Décembre 2008

[3] "Dynamic Investigation of Static Divergence: Analysis and Testing"Jennifer Heeg,Langley Research Center, Hampton, Virginia , National Aeronautics and Space Administration

[4] "Numerical Simulation of the Divergence of A Wind Turbine Airfoil_Part 1"D.Ramdenee, Ion Sorin Minea, T. D'Hamonville, A. Ilinca, LREE/UQAR, March 2010

[5] D. Ramdenee, A. Ilinca. An insight into computational fluid dynamics.
Rapport Technique , UQAR/LREE-*, *

[6] T. Tardif d'Hamonville, A. Ilinca Modélisation et Analyse des Phénomènes Aéroélastiques pour une pale d'Éolienne
Master s Thesis , UQAR/LREE-*, *

[7] D.Ramdenee, Ion Sorin Minea T. Tardif d'Hamonville, A. Ilinca Numerical Simulation of the divergence of a wind turbine Airfoil_part 1
CSME 2010, UQAR/LREE-*, *