



Numerical investigation of the structural integrity of a flexible plate considering the fluid-structure interaction

Nacim MELLEL *, Brahim MOHAMMEDI

Centre de Recherche Nucléaire de Birine, BP 180, Ain Oussera, Djelfa, Algeria

ARTICLE INFO

Article history:

Received 12 February 24

Accepted 08 March 24

Keywords:

Flexible plate; Structural integrity; fluid-structure interaction; elastic instability

ABSTRACT

In this study a numerical verification of the integrity of a flexible plate taking into account its interaction with the fluid (FSI) is presented. The bi-directional partitioned FSI approach is applied to a test case of 3D fluid flow at medium Reynolds number. The results obtained by coupling the ANSYS Transient Mechanics and ANSYS Fluent codes show that the results of the fluid flow velocity and pressure profiles are approximately close to those obtained by the MpCCI code. The elastic instability of the plate and the flow disturbances that can destroy the plate are predicted by evaluating the Strouhal frequency of the fluid flow and the natural frequency of the plate. The results obtained for van Mises stress and structural deformation were found to be below safety and design limits.

1 Introduction

In hydraulic and thermal power plants, many cases of turbulent fluid flow exist around square or round bodies, such as the cases of heat exchanger tubes in nuclear reactors, wind turbine blades and turbine plates in renewable energy facilities. These structural elements are built with acceptable structural integrity with respect to the safety of these facilities to avoid catastrophic failure and to ensure that they will perform their functions during reasonable use for as long as their life is expected.

Structural integrity is defined as the ability of a building element or structure to withstand an anticipated load without excessive deterioration or deformation through failure or fatigue. The structure must therefore be designed according to the criteria of ultimate material resistance to failure and allowable deformation.

For flexible structures immersed in a fluid, the flow of the fluid is commonly turbulent and often leads to unstable vibrational behavior that leads to premature deterioration by the vibratory fatigue phenomena, especially at their supports [1]; the problem that concerns the flow around flexible structures is therefore the vibration induced by the vortices.

For fluid flow with very low Reynolds number, the streamlines of the resulting flow are perfectly symmetrical. However, as the Reynolds number increases, the flow becomes asymmetric and the so-called Kármán vortex alloys appear

* Corresponding author. Tel.: +213669936921.

E-mail address: n.mellal@crnb.dz

and become a major source of vibrational fatigue damage [2,3]. And to avoid these undesirable vibrations, it is necessary to respect the resistance criteria and the limit parameters of the flow regimes for which the critical vibratory state of the structure is determined by the knowledge of the Strouhal number which relates the frequency of detachment of the eddies [4]. A number of experimental and numerical studies have been carried out in the fields of vibration beat instability, wake effect on plate stability, and natural frequency estimation for plates exposed to fluid flow. The self-sustained flapping of a three-dimensional flexible plate in a uniform viscous flow is numerically simulated by Yu et al. [5]. A simple numerical model cantilevered plate subject to axial flow on both surfaces is constructed by Tang et al. [6, 7]. To evaluate the post-critical instability of the flow and the behavior of the structure, a finite-element model of a plate has been realized with the intention of extending the approach combining the finite element method and classical shell theory to allow the modeling of any type of curved structure subjected to turbulent flow [8]. Tayaba et al. [9] present a study in which they estimate the maximum displacement and deformation of flaps fixed to the base and positioned at right angles to the flow; they also present the estimation of drag coefficients and calculations to define flow regimes in the wake of flaps. A study conducted by Konstantinidis [10] on the dynamic response of a 2D elastic plate as a reference case, the aim has been to investigate the relationship between the plate's structural frequency and the flow vortex. The influence of material parameters and boundary conditions on the plate's dynamic response under near-resonance conditions is also investigated, with the aim of elucidating the nature of flow-induced vibrations. Wang et al. [11] present a numerical investigation using an immersed boundary method to study the vibratory behavior of a flexible plate fixed vertically in a laminar boundary layer above a rigid wall, where several parameters such as the Reynolds number, structure-fluid mass fraction and flexural stiffness have been taken into account.

In this study, a 3D simulation of the bidirectional fluid-structure interaction of a flexible plate that can represent the case of the turbine plate or a wind turbine blade is presented. The objective is to verify the physical integrity, and geometric stability of the plate, firstly by analyzing the dynamic behavior of the turbulent flow disturbed by the plate using the velocity and pressure fields while taking into account the bidirectional fluid-structure interaction; and secondly by studying its mechanical behavior by numerically studying the evolution of stress and strain of the considered structure [12]. The existence of the vortex induced vibration (VIV) which is an important source of fatigue damage is verified by comparing the natural frequency of the emerged plate to the Strouhal frequency for the chosen test case [13].

2 Theoretical formulation

2.1 Theoretical equations of incompressible fluid mechanic

The incompressible and isothermal flow of a Newtonian fluid is governed by conservation of mass equations in differential form:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

The Navier-Stokes equations are [14,15]:

$$\rho \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} \quad (2)$$

Where $\mathbf{u}$ is the vector velocity, ρ is the density of the fluid which is constant, p is the pressure, μ is the dynamic viscosity of the fluid, ν is the kinematic viscosity of the fluid and t is time.

To solve the equations (2), an approach is used to avoid the problem of the convective term which presents a non-linearity related to turbulence; the approach consists to split the quantities present in the flow in two contributions, that of the averaged flow and that of the fluctuations related to the turbulence phenomenon. This formulation is described by the equations [14,15]:

$$\mathbf{u}(\mathbf{x}, t) = \bar{\mathbf{u}}(\mathbf{x}, t) + \mathbf{u}'(\mathbf{x}, t) \quad (3)$$

$$p(\mathbf{x}, t) = \bar{p}(\mathbf{x}, t) + p'(\mathbf{x}, t) \quad (4)$$

Once these expressions are substituted into the original Navier-Stokes equations, the unsteady averaged Navier-Stokes equations, commonly referred to as U-RANS in the literature (Unsteady Reynolds Averaged Navier-Stokes), are obtained and are represented by the equations [14,15]:

$$\nabla \cdot \bar{\mathbf{U}} = 0 \quad (5)$$

$$\frac{\partial \bar{U}_i}{\partial t} + \bar{U}_j \frac{\partial \bar{U}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_i} + \nu \frac{\partial^2 \bar{U}_i}{\partial x_i \partial x_j} - \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} \quad (6)$$

The last term of the last equation presents six additional unknowns in the system; the introduction of a turbulence model allows the implementation of additional equations to close this new system of equations. A wide variety of more or less sophisticated turbulence models exist; such k- ϵ , k- ω , and SST models that are commonly used for the prediction of fluid flow behavior.

2.2 Linear behavioral equations of solid mechanics

The dynamic behavior of a structure subject to external forces and that has small amplitude deformation, is determined by the equation as follows [14]:

$$[M]\{\ddot{\mathbf{d}}\} + [C]\{\dot{\mathbf{d}}\} + [K]\{\mathbf{d}\} = \{\mathbf{f}_{\text{ext}}\} \quad (7)$$

Where $\{\mathbf{d}\}$ is the displacement vector of the nodes of the structure. $[M]$, $[C]$ and $[K]$ are defined respectively as the mass, damping and stiffness matrices of the structure and $\{\mathbf{f}_{\text{ext}}\}$ is the vector of external forces applied to the structure by the fluid.

2.3 Interfacial boundary condition

FSI establishes the mutual interaction of a structure with a circulating fluid, and develops the dependence of their movements on each other. This type of problem is related to the effects of fluid velocity and pressure, which affect the deformation of the structure [9].

The coupling of the fluid-structure medium is modeled by the equations that establish the continuity of the normal components of the stress tensor σ and the velocity field $\mathbf{v}$ at the fluid structure interface Γ_c . Boundary conditions in this interface are summarized by:

$$\sigma_{fn} = \sigma_{sn} \quad (8)$$

$$\mathbf{v}_f = \mathbf{v}_s \quad (9)$$

Where $\mathbf{v}_s$ and $\mathbf{v}_f$ are the velocities of the structure and fluid respectively. Dynamic stability is achieved by continuity of normal stresses in the solid σ_{sn} and fluid σ_{fn} interface. $\mathbf{n}$ is the unit normal vector [14].

3 FSI simulation with ANSYS-Fluent

To model the fluid-structure coupling phenomenon, the theoretical equations given above are discretized and solved by the ANSYS-Fluent software. The method consists in solving simultaneously the equations of fluid mechanics and the equations of structural mechanics and in using a coupling system between the fluid flow solver and the mechanical solver. For each iteration, pressure and displacement data are transferred between the two modules to perform the simulation including the fluid-structure interaction.

The data transfer steps performed by the coupling system are as follows [14]: The Fluent fluid code transmits the nodal pressures of the interface to the ANSYS Mechanical structural code, the Mechanical code computes the solution and updates the node coordinates and transmits the new positions of the interface nodes, the Fluent code updates the mesh, taking into account the new node positions and computes the pressure solution; return to step 1.

3.1 Simulation of the flexible plate in a fluid channel considering the FSI

The objective of this study is to predict the behavior of a flexible plate by considering the 3D simulation of the bi-directional fluid-structure interaction by using the most advanced coupling techniques that the ANSYS code offers.

The simulation of the test case [13] of turbulent flow in a channel perturbed by an obstacle of a square plate is performed using the k- ϵ turbulence model and delimiting the study domain by the boundary conditions that well describe

the problem. The fluid is considered incompressible and the solid is an elastic solid body. The current simulation is based on two main numerical methods: the Finite Element Method (FEM) and the Finite Volume Method (FVM).

3.1.1 Simulation parameters and coupling procedure

3.1.1.1 Geometric model

The dimensions of the plate and the flow parameters are taken from the test case of Vesa Nieminen [13]. The fluid channel is a parallelepiped of length $L=0.25\text{m}$ and height $h=0.052\text{m}$ (Fig. 1). The flow is assumed to be stationary, three-dimensional, incompressible, viscous, and without heat transfer. Flow velocity is $U=8\text{m/s}$ and the angle of flow incidence is 15° . The geometry of the plate is the square section: side= 0.048m and thickness 0.002m , the Young's modulus is $E=5 \times 10^5\text{ Pa}$ (Fig. 1).

3.1.1.2 Domain boundary conditions

The fluid boundary conditions are as follows:

- inlet velocity,
- outlet pressure,
- The upper and lower boundaries: wall,
- Profile and flaps: wall,
- Plate boundaries: fluid structure interface (06 faces).

The boundary conditions of the plate: A fixed end and the external part are considered as a fluid structure interface.

3.1.1.3 Meshing

The modeling of the structural domain is performed in the ANSYS Workbench environment. The plate domain is meshed with structured mesh (Fig. 2). The mesh is well refined around the plate obstacle to capture the basic flow characteristics. Ansys Fluent is used to mesh the fluid domain. The mesh is a 3D unstructured type; it composed of tetrahedral elements. A dynamic mesh was applied to the fluid mesh at the vicinity of the interface with the flexible plate. The remeshing and smoothing method was successfully applied to improve the accuracy of the results. The dependency of the calculated physical parameters on grid size is achieved, several densities of grid were tested to obtain the appropriate one which ensures minimum uncertainties; for which there is almost no change in the velocity profile when the mesh is more refined [16]. A difference of 1% between two consecutive results [17] of calculated velocities is considered as the selection criterion of the adopted grid. The total number of cells considered is 51490 (Fig. 2).

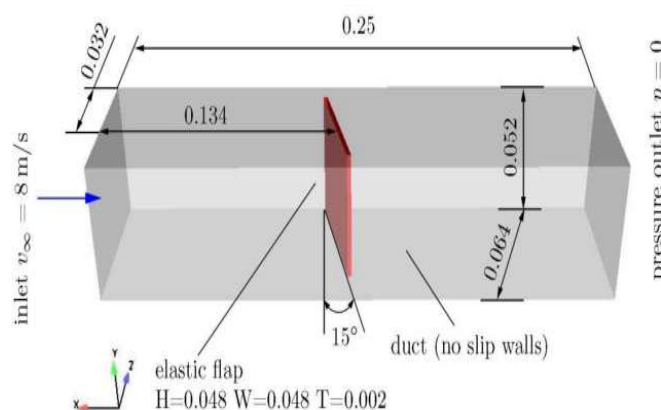


Fig. 1 –Test case of a plate surrounded by a fluid domain [13]

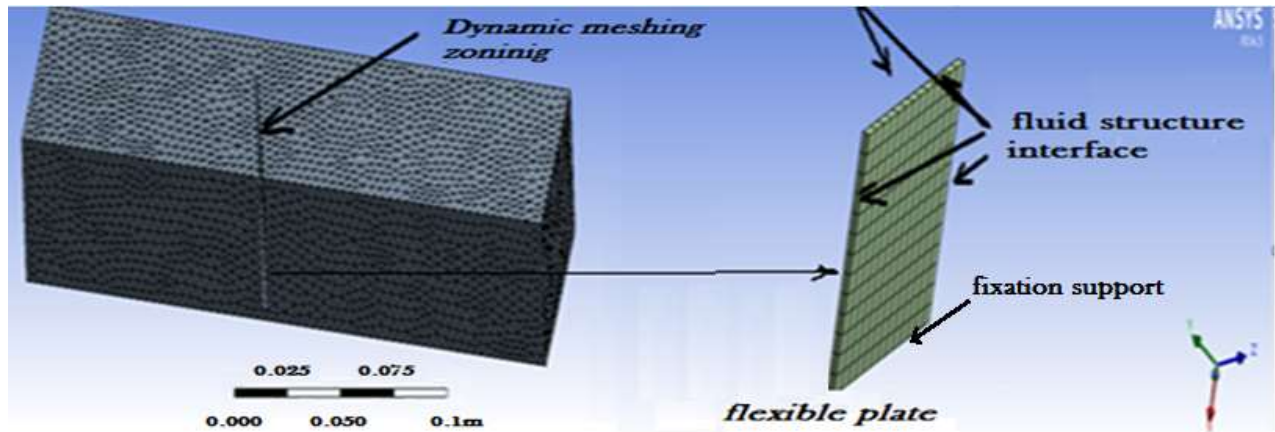


Fig. 2 –Fluid domain meshing and flexible plate interface

4 Results and discussion

In this study, we applied a bidirectional partitioned approach by coupling ANSYS Mechanical and Fluent codes using the $k-\epsilon$ turbulence model. The objective is to predict the vibration displacement of the plate and flow characteristics such as velocity component profile and pressure distribution. The plate displacement is shown in Fig. 3; the stress distribution of the structure is given in Fig. 4.

To evaluate the structural integrity of the plate, the stress and strain distribution is considered [12], the strain contour shows the existence of a localized high strain region with a value of 0.65 mm (Fig. 3), such high strain occurs at the free section of the flexible plate. The stress contour in Fig. 4 reveals that the section that is close to the fixed side of the plate has a higher stress. However, the maximum stress in the plate is identified equal to 1.40 MPa, which is significantly lower than the yield strength of the plate material which is 10.48 MPa [18]. Therefore, regular maintenance and inspection should be performed especially in this region for any signs of crack propagation. Therefore, the plate would not fail, at the current state of the plate loading conditions described by the fluid flow conditions while considering the fluid-structure interaction.

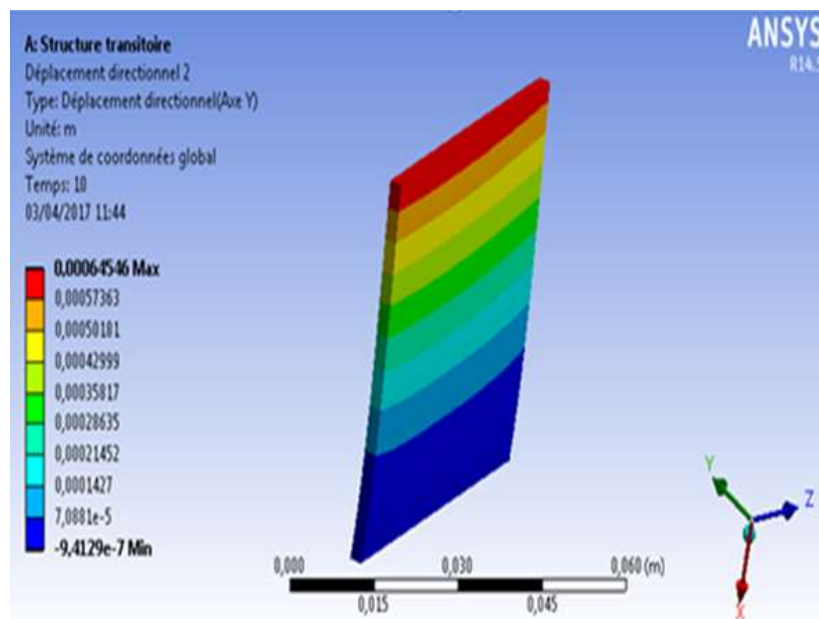


Fig. 3 –Oscillation displacement of the plate in y direction

Figures 5 and 6 below, show respectively, velocity profiles, as well as the pressure profiles around the square plate at different x positions and for $z=0.030$ m plane. The velocity results obtained show that they are reasonably well within the range of those obtained by the MpCCI code used in the test case. [13].

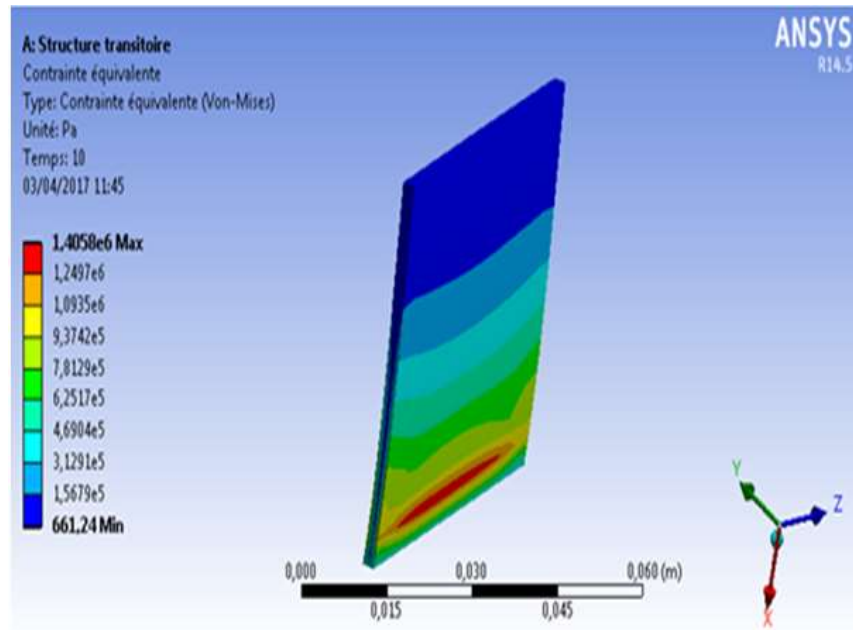


Fig. 4 –Van Misés stress distribution in the plate

The pressure profiles (Fig. 6), the velocity contour with streamlines in the fluid domain (Fig. 8) and the evolution of the velocity v versus the fluid domain length (Fig. 9) are established using the ANSYS Fluent solver. The examination of these contours and velocity streamlines of the flow at several points showed that around the perpendicular plate, the pressure values are maximum and the velocity profile presents an asymmetry at the outlet of the channel (Fig. 6 and Fig. 7), this asymmetry revealed by the existence of small vortices that can give rise to vortex-induced vibrations of the plate. The value of the Reynolds number for the characteristics of the flow channel configuration (velocity and thickness of the plate) compared to the critical value of Re obtained experimentally confirms that the flow regime around the plate is turbulent considered as subcritical mode [19], this is why the study of vortex induced vibration is conducted to verify the integrity of the plate against this phenomenon.

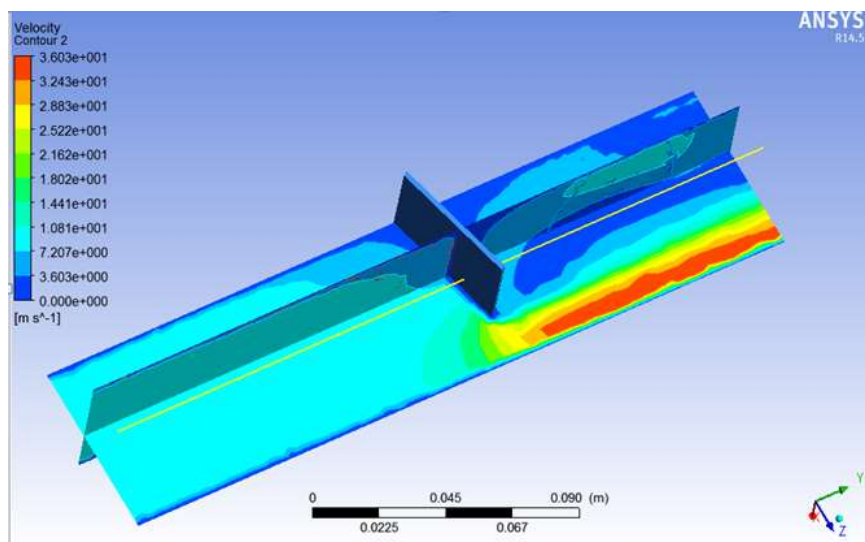


Fig. 5 –Velocity distribution in fluid domain

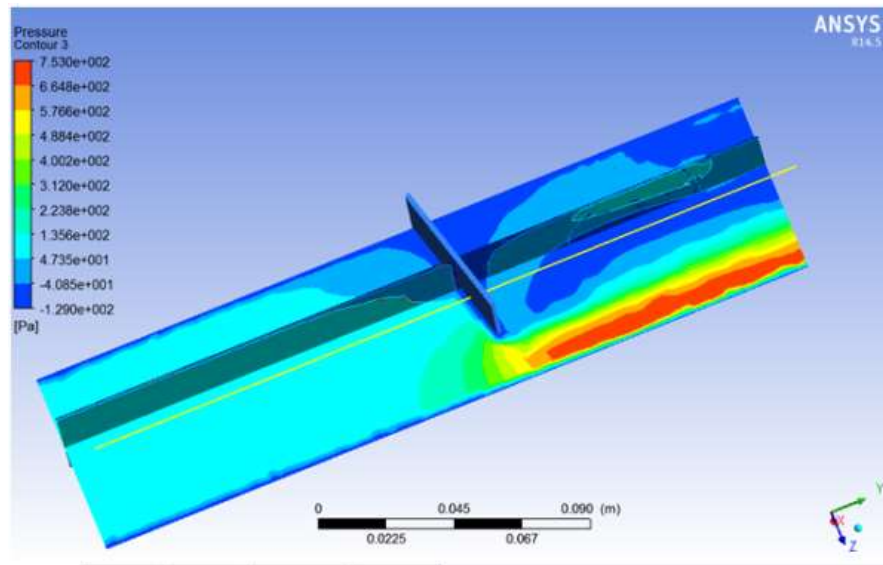


Fig. 6 – Pressure contour of the fluid domain

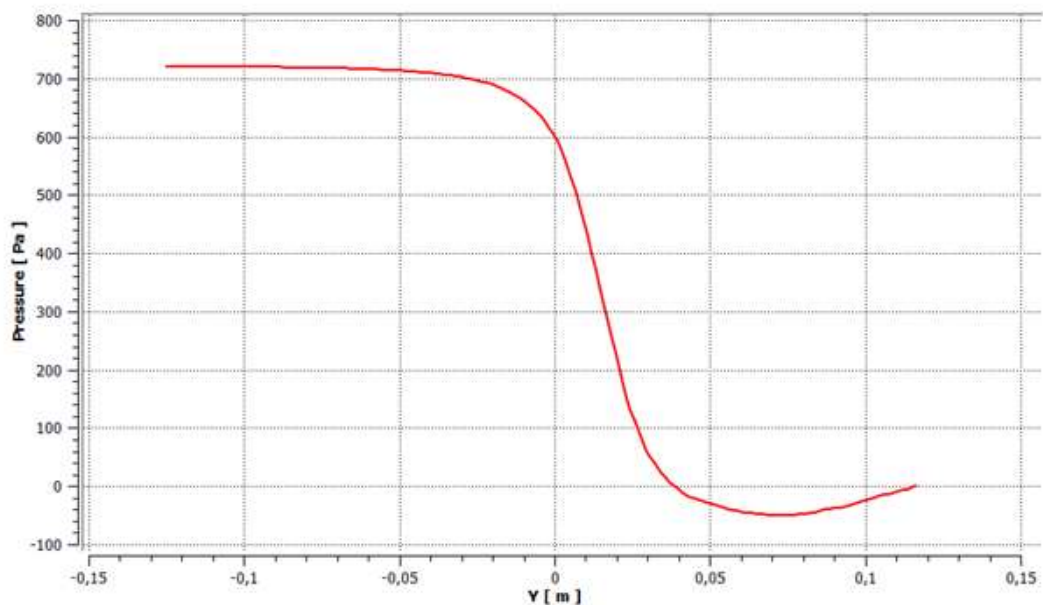


Fig. 7 –Pressure evolution versus the length of the fluid domain

In most cases, the strong vibration created by the formation and loss of vortices in the flow wake around thin structure can produce fatigue damage. This vibration is generated by alternating forces whose frequencies are determined by the non-dimensional Strouhal number. This vibration become damaging as soon as the loss frequency of the vortices approaches the natural frequency of the body, a phenomenon called frequency coupling. The study of this undesirable phenomenon has major importance for the design and operational safety of structure surrounded by fluid. In engineering, fluid forces and Strouhal numbers are the main factors to be considered in the design of slender structures subjected to transverse flow.

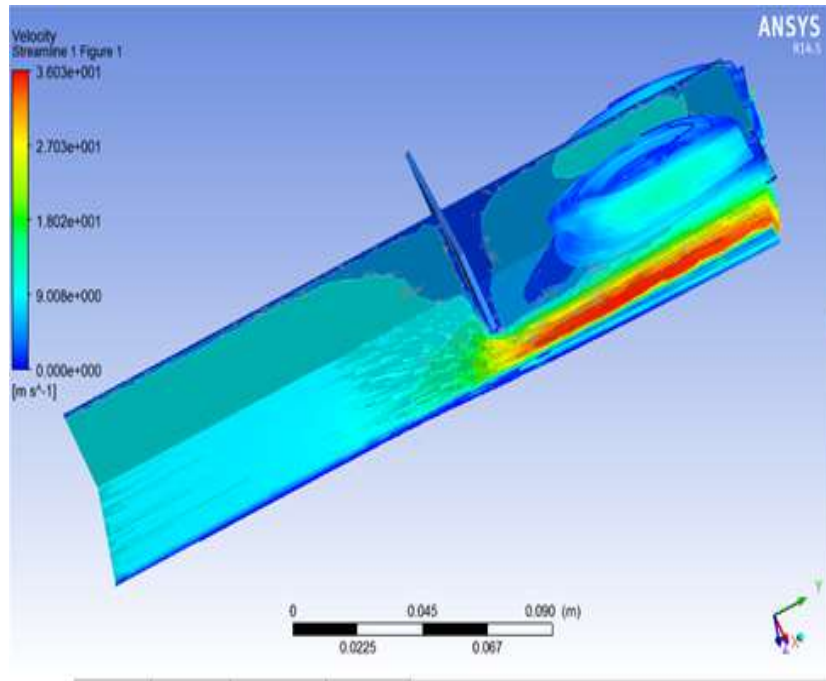


Fig. 8 - Velocity contour with streamlines in the fluid domain

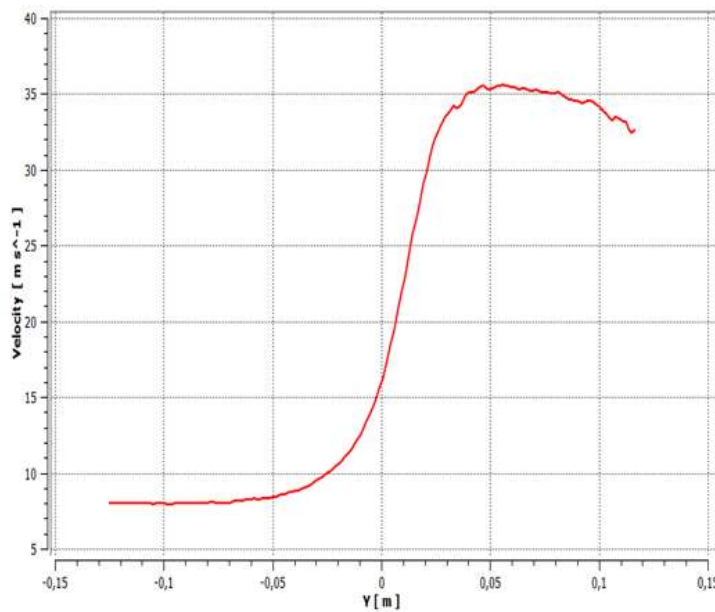


Fig. 9 –Evolution of the velocity v versus fluid domain length

The first three bending modes of the plate are determined numerically by modal analysis; they are given respectively for the first three bending modes by values 184.65 Hz, 1143.6 Hz and 3168 Hz (fig.10).

The Strouhal number is calculated by the following equation:

$$S_t = (fL/U) \sin \alpha \quad (10)$$

Where L is the characteristic length (airfoil thickness = 0.002m), U is flow velocity, f is the vortex release frequency and α is angle of attack to a main flow stream.

The shedding frequency of the upper and lower vortices was measured experimentally in the cases of the similar submerged plate. It was found that for this flat plate, the Strouhal number for both upper and lower vortices first increases,

reaches a maximum at $\alpha = 15^\circ$ and then decreases (Fig. 11). A sudden jump in the value of the Strouhal number could be noted between $\alpha = 0^\circ$ and $\alpha = 5^\circ$.

Using the limited value of Strouhal number S_t of 0.2 for the case of the plate with angle of attack of 15° for a wide range of flow velocities [20]; the value of the shedding frequency is therefore found closer to 800 Hz. Comparing this frequency with the natural frequency of the plate which is 184.65 Hz (Fig. 10), it was found that the Strouhal frequency is far from the value of the natural frequency of the plate. It can be concluded that the phenomenon of elastic instability induced by the flow is far from being reached; the risk of amplification of oscillations that could damage the plate is therefore minimal.

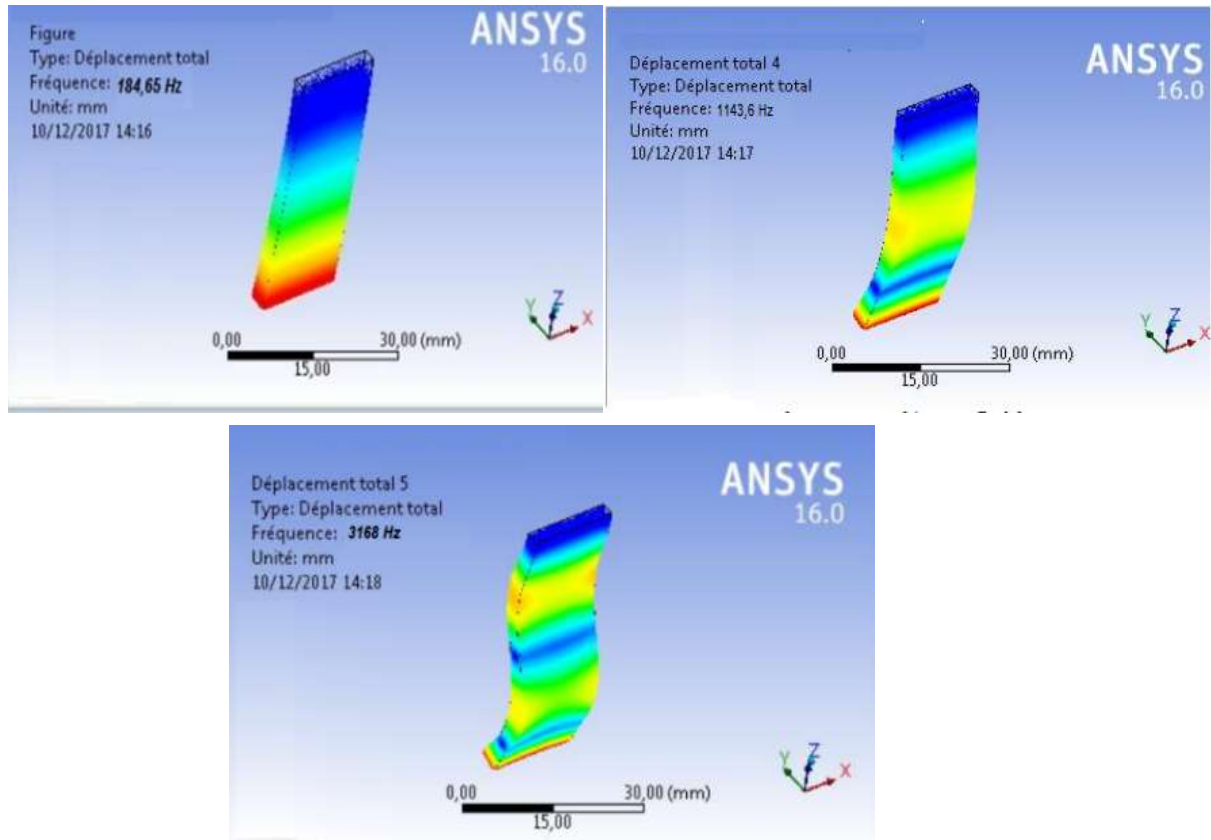


Fig. 10 –First three bending modes of the plate

5 Conclusion

The assessment of the structural integrity of a flexible plate considering fluid-structure interaction with a flow incidence angle was the objective of current research. The numerical simulation 3d by ANSYS Fluent was adopted and validated with data from the MpCCI code applied for a test case of a Newtonian and incompressible fluid flow in turbulent regime around a solid like plate. This study has allowed us to get some observations about the coupling success between the codes Fluent and ANSYS Transient Mechanics. Moreover, the current and pressure lines are determined, flow velocities are found to be far from values that cause the dynamic instability that can induce the ruin of the flexible plate. The obtained results of van Mises stress and the amplitude of oscillation of the considered flexible plate considering the fluid-structure interaction are lower than the design limit values which confirms the structural integrity of the plate for the operating conditions of the chosen test case.

REFERENCES

- [1] Yoann J., Modélisation et simulation numérique de vibrations induites par écoulements autour d'obstacles cylindriques seuls ou en réseaux, PhD thesis, Université Pierre et Marie Curie, Paris, 2011.
- [2] Moudid L.A., Couplage fluide structure pour la simulation numérique des écoulements fluides dans une conduite à parois rigides ou élastiques en présence d'obstacles ou non, Sciences de la Terre, PhD thesis, Université d'Artois, 2007.
- [3] Lam K.M., Leung M.Y.H., Asymmetric vortex shedding flow past an inclined flat plate at high incidence. *Eur. J. Mech. B*. Vol. 24 (2005) pp.33-48. <https://doi.org/10.1016/j.euromechflu.2004.05.004>
- [4] Blevins R.D., Flow induced vibration, Van Nostrand, Reinhold Inc, 2nd Edition, 1990.
- [5] Yu Z., Wang Y., Shao X., Numerical simulations of the flapping of a three-dimensional flexible plate in uniform flow. *J. Sound Vib.* Vol.331 (2012) pp.4448-4463. <https://doi.org/10.1016/j.jsv.2012.05.009>
- [6] Tang L., Païdoussis M.P., On the instability and the post-critical behavior of two-dimensional cantilevered flexible plates in axial flow. *J. Sound Vib.* Vol.305 (2007) pp.97-115.
- [7] Tang L., Païdoussis M.P., The influence of the wake on the stability of cantilevered flexible plates in axial flow. *J. Sound Vib.* Vol.310 (2008) pp.512-526. <https://doi.org/10.1016/j.jsv.2007.03.042>
- [8] Kerboua Y., Lakis A., Thomas M., Marcouiller L., Vibration analysis of rectangular plates coupled with fluid. *Appl. Math. Model.* Vol.32 (2008) pp.2570-2586. <https://doi.org/10.1016/j.apm.2007.09.004>
- [9] Tayaba B., Franziska H., Martin H., Ruediger S., Investigation of fluid-structure interaction induced bending for elastic flaps in a cross flow, *Appl. Sci.* Vol.10 (2020). <https://doi.org/10.3390/app10186177>
- [10] Konstantinidis E., Cross-Flow-Induced Vibration of an Elastic Plate. *Fluids*, Vol.6, 82 (2021). <https://doi.org/10.3390/fluids6020082>
- [11] Wang L., Lei C., Tian F., Fluid structure interaction of a flexible plate vertically fixed in a laminar boundary layer over a rigid wall. in: *Proceedings of the 21st Australian Fluid Mechanics Conf.*, Australia, 2018.
- [12] Mellel N., Mohammedi B., et al., Structural integrity evaluation of irradiated LEU targets for the production of molybdenum-99 using thermo-mechanical behavior simulation coupled with pressure of fission gas release calculation. *Nucl. Sci. Tech.* Vol.33, 152 (2022). <https://doi.org/10.1007/s41365-022-01143-6>
- [13] Nieminen V., Fluid structure interaction simulation utilising MpCCI. VTT technical research centre of Finland. VTT research report VTT-R-01447-15, (2015). <https://publications.vtt.fi/julkaisut/muut/2015/VTT-R-01447-15.pdf>
- [14] Dompierre F., Détermination des fréquences naturelles de structures submergées par la méthode d'interactions fluide structure bidirectionnelle, Mémoire du diplôme de maîtrise sciences appliquées, Université de Montréal, Canada, 2010. https://publications.polymtl.ca/302/1/2010_FrederickDompierre.pdf
- [15] Oudran A., Aour B., Numerical investigation of a laminar flow in a tubular chain of a heated floor slab. *Recueil de mécanique (RDM)*, Vol.2 (2) (2017) pp. 192-203. DOI : 10.5281/zenodo.1173687
- [16] Rahmani Z., Kahil Y., Benlefki A., Numerical study of turbulent flow around four cylinders in square configuration, *Recueil de mécanique (RDM)*, Vol.4 (2) (2019) pp. 359-373. DOI: 10.5281/zenodo.3738717
- [17] Salhi M., Mohammedi B., Laouar S., Conceptual design of irradiation device for silicon neutron transmutation doping around Es-Salam research reactor. *Nucl. Sci and Tech.*, Vol.31 (38) (2020). <https://doi.org/10.1007/s41365-020-0747-3>
- [18] Qiang X., Minglei P., Lixia Z., Yuanyuan Z., Shengyu F., Mechanical properties of silicone rubber composed of diverse vinyl content silicone gums blending, *Mater. Des.* Vol.31 (2010) pp.4083-4087. <https://doi.org/10.1016/j.matdes.2010.04.052>
- [19] Lam K.M., Leung M.Y.H., Asymmetric vortex shedding flow past an inclined flat plate at high incidence, *Eur. J. Mech. B*. Vol.24 (2005) pp.33-48. <https://doi.org/10.1016/j.euromechflu.2004.05.004>
- [20] Damu M., Balu H.K.H., Arunkumar K., Ajith K. R., Flow past a curved plate: wake vortex characteristics, in: *Proceedings of the 2nd International Conf. for Convergence in Technology (I2CT)*, 2017. Mumbai, India.