10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE AIAC-2019-170
18-20 September 2019 - METU, Ankara TURKEY

FLUTTER ANALYSIS AT TRANSONIC SPEEDS USING A COUPLED CFD-CSD SOLVER

Emre Alpman[1]
Marmara University,
Istanbul, Turkey

Levent A. Kavurmacıoğlu[2]
Istanbul Technical University
Istanbul, Turkey

Seher Eken[3]
Samsun University
Samsun, Turkey

ABSTRACT

In this paper, the CFD/CSD coupled solver based on open-source software is developed to simulate and predict the transonic flutter for the aeroelastic system. The pressure based OpenFOAM solver sonicDyMFoam is used for the CFD simulation. This aerodynamic solver is then coupled with an OpenFoam structural solver sixDoFRigidBodyMotion. The results of the unsteady aerodynamic flow and transonic flutter simulations are presented for the selected NACA 64A010 airfoil.

INTRODUCTION

Aeroelasticity is a multi-disciplinary field, focusing on the interaction of inertial, structural and aerodynamic forces. In classical theories of aeroelasticity, aerodynamic and structural forces are assumed to be linear. For several decades, the classical approaches have been widely used to estimate the flutter speed and frequency of the linear structure. However, they fail to capture the phenomena resulted from structural and aerodynamic nonlinearities [Eken and Kaya, 2015].

Aerodynamic nonlinearities are often encountered at transonic speeds or high angles of attack where flow separation occurs. When the flight speed exceeds Mach 0.7, the flow around the aircraft exhibits shock waves and highly nonlinear behavior due to the transition between subsonic flow and supersonic flow. The linear aerodynamic theories fail to capture the flutter in this transonic regime which is known to be the most critical flutter speed amongst all regimes, also called as ‘transonic dip’. Recently, due to the developments in computing technologies, more sophisticated tools have been applied for simulation and prediction of the transonic flutter. These tools are generally based on the coupling of computational fluid dynamics (CFD) and computational structural dynamics (CSD) codes to obtain the aeroelastic response of the structure with nonlinear aerodynamics. Using CFD/CSD tools, several studies have been conducted to predict the flutter [Gao et al. 2005, Opgenoord et al. 2017, Alonso and Jameson 2018] of the aeroelastic system.

The current research interest in transonic aeroelasticity has grown progressively due to its essential role in the design of modern civil aircrafts/jets design. By the increased computing power one can enable the CFD/CSD codes to predict the transonic aeroelastic behavior of

[1] Assoc. Prof. Dr. in Mechanical Engineering Department, Email: emre.alpman@marmara.edu.tr
[2] Assoc. Prof. Dr. in Mechanical Engineering Department, Email: kavurmacio@itu.edu.tr
[3] Asst. Prof. Dr. in Aerospace Engineering Department, Email: seher.eken@samsun.edu.tr

the aircraft and thereby reducing the number of expensive wind-tunnel tests in the design of a supersonic aircraft. Within this context, the objective of this work is to develop an accurate and efficient CFD/CSD solver to predict the transonic flutter of the typical-section wing model. The CFD part of the code based on an solver which is suitable for aeroelastic problems is adapted from the most widespread open source software, OpenFOAM transonic solver *sonicDyMFoam*. The CSD part of the code is developed also using an OpenFoam structural solver sixDoFRigidBodyMotion. To test the accuracy of the solver, the airfoil NACA 64A010 is chosen and the transonic flow simulations are performed.

GOVERNING EQUATIONS

Aerodynamic Model

In this study, the unsteady two-dimensional flow of a viscous, compressible gas is considered. The governed system of such flows is given by the Navier-Stokes equations. At transonic speeds the Reynolds number of the flow will be quite high. Therefore, the flow field is assumed to be fully turbulent and Menter's k-ω SST turbulence model. The corresponding equations are given as [Marcantoni et al. 2012]:

- Conservation of Mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}) = 0 \tag{1}$$

- Conservation of momentum (body forces are neglected):

$$\frac{\partial (\rho \boldsymbol{U})}{\partial t} + \nabla \cdot (\rho \boldsymbol{U}\boldsymbol{U}) = \nabla \mathrm{p} + \nabla \cdot \hat{\boldsymbol{\tau}} \tag{2}$$

- Conservation of energy:

$$\frac{\partial (\rho E)}{\partial t} + \nabla \cdot [\boldsymbol{U}(\rho E)] + \nabla \cdot \mathbf{q} + \nabla \cdot [(p\boldsymbol{I} - \hat{\boldsymbol{\tau}})\boldsymbol{U}] = 0 \tag{3}$$

where ρ is the mass density, U the fluid velocity and p the pressure. In addition, τ is the viscous stress tensor and given as

$$\hat{\boldsymbol{\tau}} = 2\mu \left[\widehat{\boldsymbol{D}} - \frac{1}{3}\mathrm{tr}\big(\widehat{\boldsymbol{D}}\big)\hat{\boldsymbol{I}}\right] \tag{4}$$

where μ is the dynamic viscosity, $\widehat{\boldsymbol{D}}$ is the deformation gradient tensor and $\hat{\boldsymbol{I}}$ is unit tensor

Aeroelastic Model

In order to perform flutter analyses we use the typical-section wing model which is sketched in Figure 1. This is based on a rigid, elastically-restrained, two-dimensional wind tunnel wing model. The translational and rotational springs reflect the bending and torsional stiffnesss of the wing. These linear springs are attached to the airfoil section at its elastic axis. Also from this figure, the bending and torsional motions of the airfoil about the elastic axis are represented by h and α, respectively.

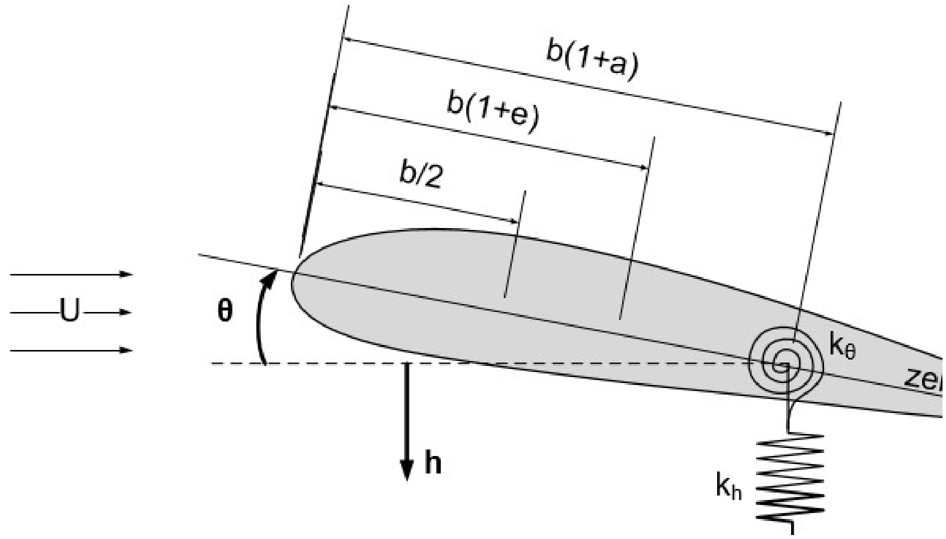


Figure 1*:* The typical-section wing model

The aeroelastic system is established using this wing model featuring pitching and plunging motion and the governing equations of motion corresponding this system is written as

$$m\ddot{h} + mbx_\theta\ddot{\theta} + k_h h = -L \tag{5a}$$

$$I_P\ddot{\theta} + mbx_\theta\ddot{h} + k_\theta\theta = M_{\frac{1}{4}} + b\left(\frac{1}{2} + a\right)L \tag{5b}$$

where the dimensionless parameter that represents the offset of the center of mass from the reference point is $x_\theta = e - a$ and the airfoil semi-chord is b. The wing having mass m, a moment of inertia I_P is elastically restrained with linear springs with constants k_h and k_θ. Also, L and $M_{1/4}$ are the positive up lift and positive nose up moment about the quarter chord.

The aeroelastic equations are non-dimensionalized using the following dimensionless quantities:

$$\tau = \omega_\theta t \qquad \omega_\theta = \sqrt{\frac{k_\theta}{I_P}} \qquad \omega_h = \sqrt{\frac{k_h}{m}}$$

Using these, Eqs. (5a-b) are rewritten in matrix form as:

$$\begin{bmatrix} mb^2 & mb^2x_\theta \\ mb^2x_\theta & I_P \end{bmatrix}\begin{Bmatrix} \ddot{h}/b \\ \ddot{\theta} \end{Bmatrix} + \begin{bmatrix} mb^2\omega_h^2 & 0 \\ 0 & I_P\omega_\theta^2 \end{bmatrix}\begin{Bmatrix} h/b \\ \theta \end{Bmatrix} = \begin{Bmatrix} -\acute{L} \\ \acute{M} \end{Bmatrix} \tag{6}$$

SOLUTION METHODOLOGY

For the CFD part, the equations of highly nonlinear transonic flow is solved using the pressure based solver called sonicDyMFoam [https://www.openfoam.com/, https://openfoam.org/]. The flow field is assumed to be fully turbulent and Menter's k-ω SST turbulence model. Here, since the computational mesh used is not fine enough to resolve the boundary layer, wall functions are used on solid surfaces. The sonicDyMFoam solver is capable of simulating the unsteady transonic, turbulent flow of a compressible gas, with optional mesh motion.

For the CSD part, one has to solve the translational and rotational motion of the rigid airfoil which elastically restrained about its elastic axis. To do this, an OpenFoam structural solver sixDoFRigidBodyMotion is used. This structural solver not only can handle the solution of translational and rotational movement in *x*-, *y*- and *z*- directions of a solid body, but also can be employed with the solvers using dynamic mesh. Among other methods such as Crank-Nicholson and symplectic, the Newmark method is selected to solve the system given in Eq. (6).

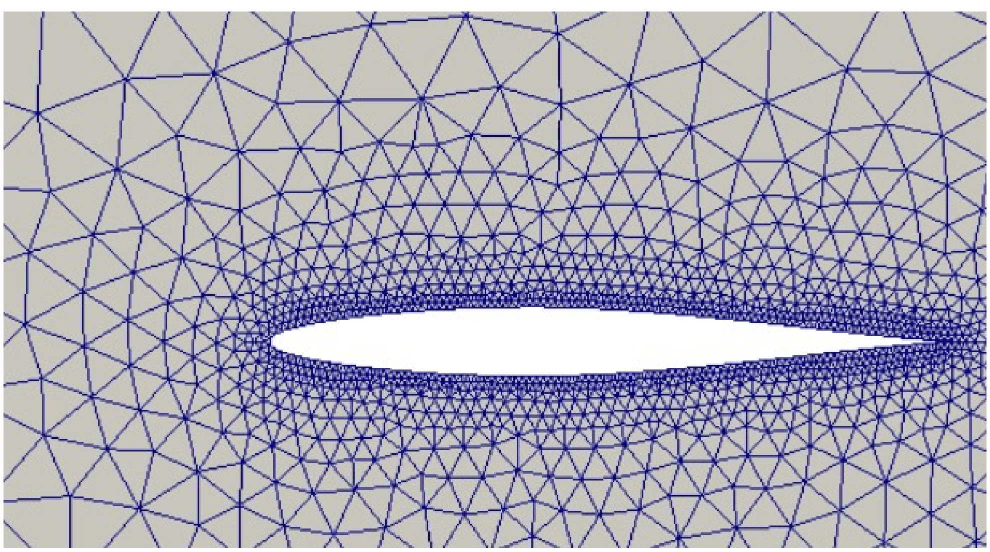
RESULTS AND DISCUSSION

To validate the developed CFD/CSD solvers a 2-D simulation of the NACA 64A010 airfoil is selected [Isogai, 1979, Fereidooni 2018]. The properties of the airfoil is given in Table 1.

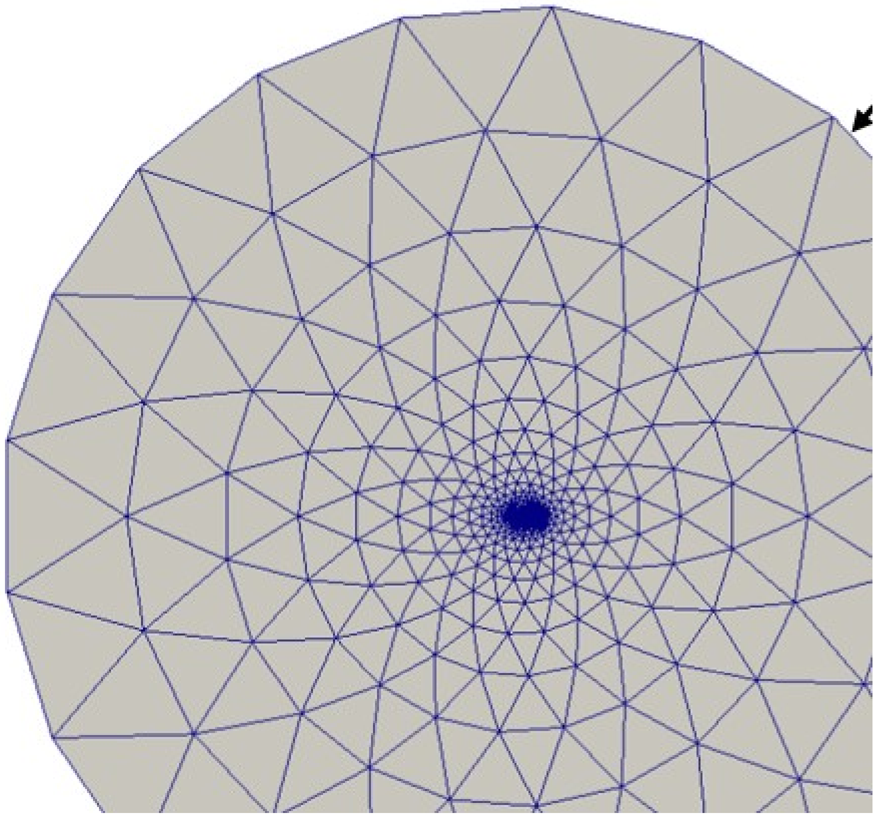
Table 1: The properties of NACA 64A010 airfoil

Mass, m	33.93 kg /m
Mass moment of Inertia, I_p	29.52 kg.m^2 /m
Translational spring stiffness, K_h	339290 N/m /m
Rotational spring stiffness, K_θ	295180 N/m /m
Uncoupled natural frequency in bending, ω_h	100 rad/s
Uncoupled natural frequency in torsion, ω_θ	100 rad/s
Mass ratio, μ	60
Half chord, b	0.45 m
Position of elastic axis from mid chord, a	-2
Position of cg from mid chord, $\boldsymbol{x_\theta}$	-0.1 m

The unstructured mesh is generated for the circular computational domain. Figure 2 shows the entire mesh domain and mesh around the airfoil. Additionally, the unstructured mesh consists of 2920 triangular cells and 3144 node.



(a)

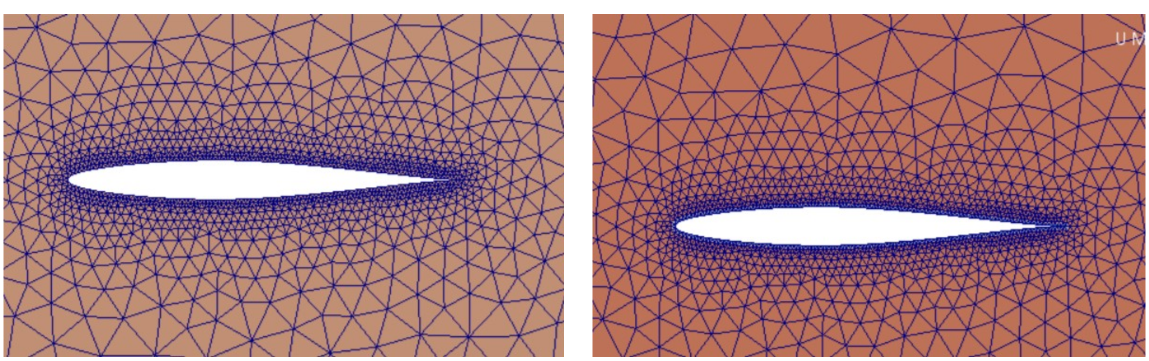


(b)

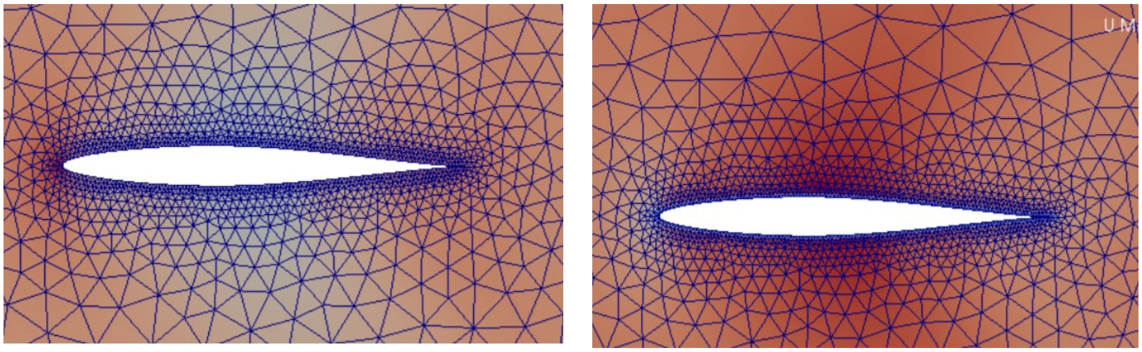
Figure 2: Mesh of NACA 64A010 (a) for the entire domain; (b) around airfoil

Case Study: Pitching Motion of NACA 64A010

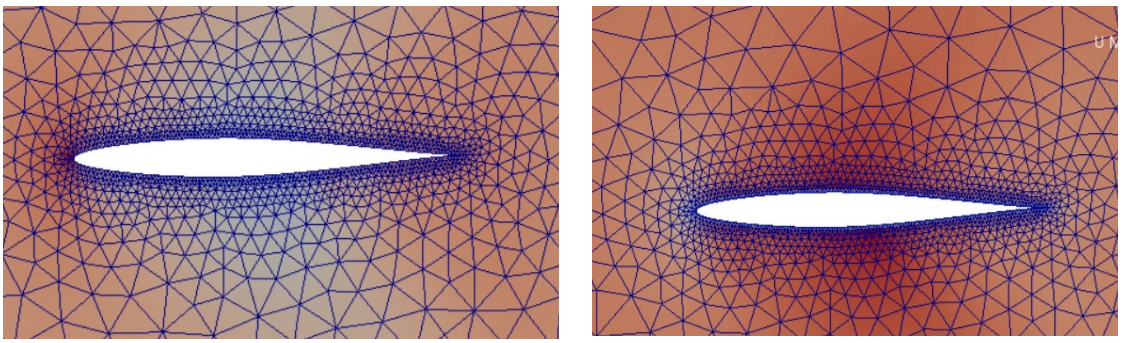
As preliminary analysis, we first tested our CFD/CSD solver to generate the pitching motion of NACA 64A010 airfoil. Initially, a Mach number of M=0.796 is given and a solution for the dimensionless time of 500 is performed where the airfoil is not allowed to move at the first second and then it is allowed to pitch for the next 4 seconds. Figure 3 shows the airfoil and pressure and velocity number contours at τ=100, 200, 300, 400 and 500.



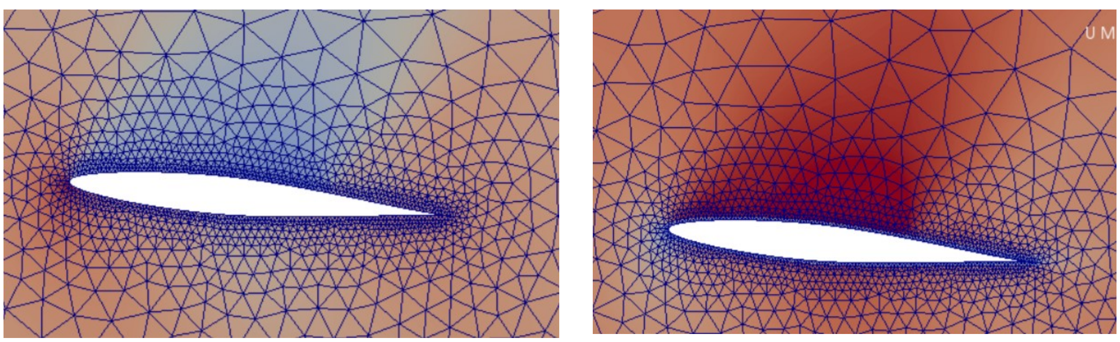
(a) τ=0



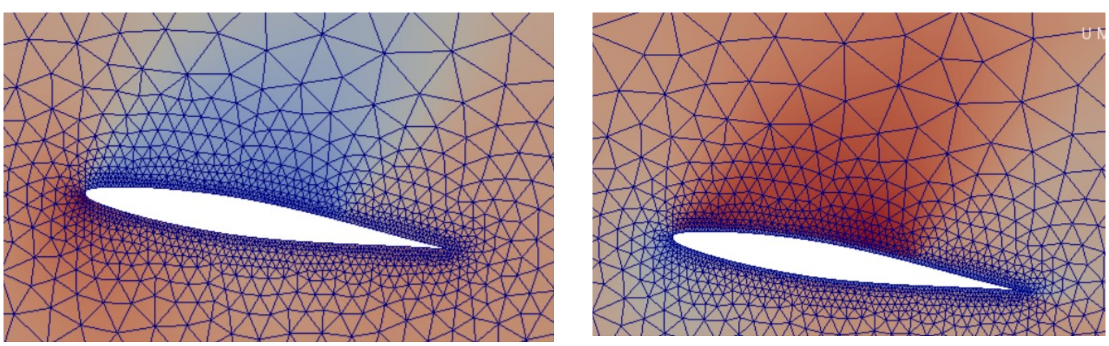
(b) τ=100



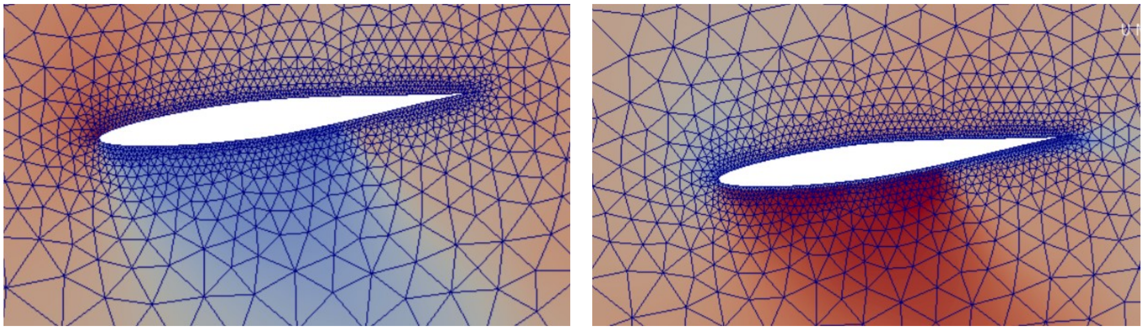
(c) τ=200



(d) τ=300



(e) τ=400



(f) τ=500

Figure 3: Pressure and velocity contours at τ=100, 200, 300, 400 and 500.

It is clear that the displacement of airfoil from its initial position increased with time. This also affects the flow field considerably.

Another result is plotted in Figure 4 which shows the time variations of the tip displacements. In order to measure the tip displacements, a point is selected near trailing edge location (see top left Figure 4).

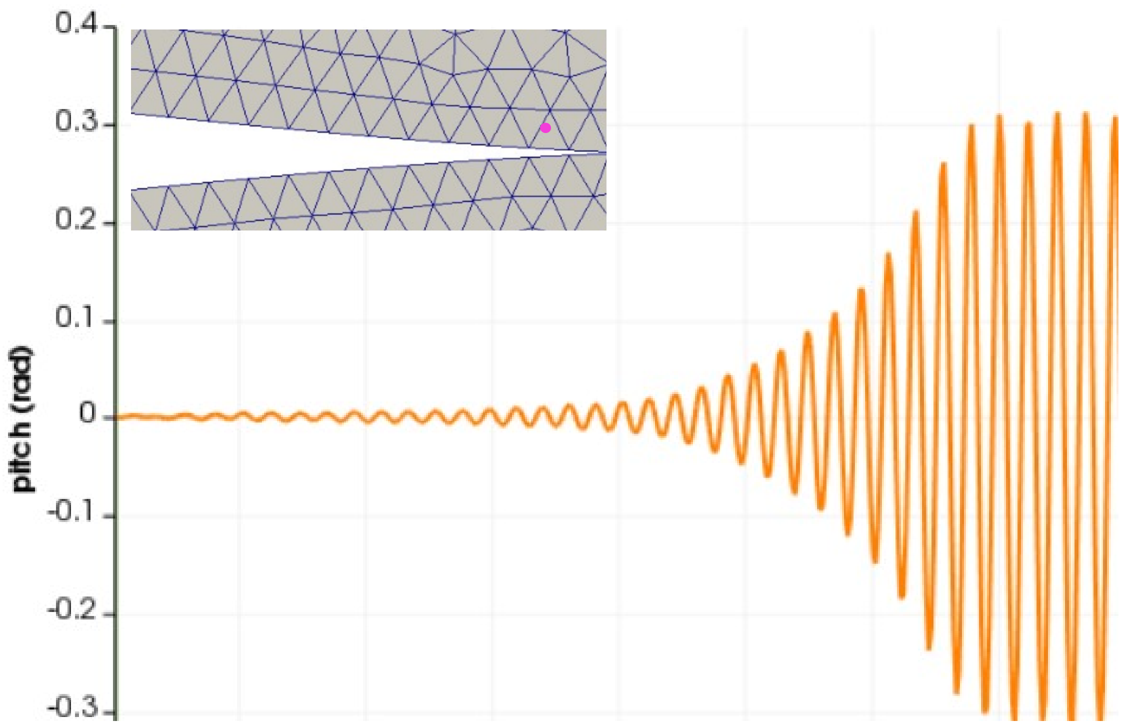


Figure 4: Tip displacements with respect to time

CONCLUSIONS

Two dimensional aeroleastic simulations were performed using the fluid-structure interaction features of the open source computational fluid dynamics software OpenFOAM. Currently, the test case of Isogai's wing [Isogai, 1979] was not be able to validated. Possible issues may be:

- Uncertainties in the calculation of wing and flow parameters
- Problems in the modelling of the dynamic mesh.

In spite of these issues the heaving plunging motion of the airfoil was able to be simulated successfully.

References

Alonso A. and Jameson A. (1994). *Fully-implicit time-marching aeroelastic solutions*, 32nd Aerospace Sciences Meeting and Exhibit, Aerospace Sciences Meetings, AIAA-94-0056.

Eken S. and Kaya M.O. (2015). *The effect of sweep angle on the limit cycle oscillations of aircraft wings,* Advances in Aircraft and Spacecraft Science, Vol. 2, No. 2, pp 199-215.

Fereidooni A., Grewal A.K. Seraj S. and Grzeszczyk M. (2018). *Computational Aeroelastic Analysis using an Enhanced OpenFOAM-based CFD-CSD Solver*, 2018 AIAA/ASCE/AHS/ASC Structures, Structural Dynamics, and Materials Conference, AIAA SciTech Forum, AIAA 2018-1955.

Gao C., Luo S., Liu, F. and Schuster, D.M. (2005). *Calculation of airfoil flutter by an Euler method with approximate boundary conditions*. AIAA Journal, 43, 295–305.

https://www.openfoam.com/

https://openfoam.org/

Isogai, K. (1979). On the transonic-dip mechanism of flutter of a sweptback wing. *AIAA Journal*, *17*(7), 793-795.

Marcantoni L.F.G., Tamagnoa J.P. and Elaskara S.A. (2012), High speed flow simulatıon using OpenFoam, Mecánica Computacional, 31, 2939-2959.

Opgenoord M.M. Drela M. and Willcox K.E. (2017) *Towards a Low-Order Model for Transonic Flutter Prediction*", 8th AIAA Theoretical Fluid Mechanics Conference, AIAA AVIATION Forum, AIAA 2017-4340.