Numerical and experimental study of flutter in an airfoil

Sinforiano Cantos Trigo[1]

Student at Universitat Politècnica de València[2]*, Valencia, Comunidad Valenciana, Spain.*

In this work it is analysed the utility of CFD[3] with FSI[4] in order to study the flutter phenomenon in wings, using a NACA 63A415 airfoil. The study includes wind tunnel testing of the airfoil in order to validate the numerical calculation. Finally, the comparison of both methods shows that the CFD study reproduces the phenomenon with similar results, making this method interesting to study flutter.

Keywords: ***aeroelasticity*, *flutter*, *aerodynamics*, *CFD*, *wind tunnel*.**

1 Motivation

The motivation of this work is to study the utility of CFD with FSI and an overset mesh to characterise the aeroelastic phenomenon of flutter in wings. It is fundamental to determine the velocity of flutter and its frequency as if it is not controlled it will cause structural failure due to the interaction between aerodynamic forces and wing deformation [1].

2 Preface

Traditionally, flutter was studied with analytical methods whose calculation times are small, with the drawback of little detail in the results; or experimental study which takes a lot of time and are really expensive.

An alternative to this methods could be CFD. However, it is necessary to model FSI, so a deforming mesh was needed, which required to be really careful in the pre-processing as many problems could arise afterwards, with the consequent cost of time. During the calculation, the mesh could deform too much, obtaining invalid results, and this is not always easy to predict.

With an overset mesh, the problem simplifies considerably, making interesting the use of this method to study flutter in wings. The overset mesh is created where the movement is allowed (near the wing) and, with similar size of elements compared to the regular mesh, results can be interpolated between both meshes.

[1] sincantr@etsid.upv.es
[2] From now: UPV.
[3] Computational fluid dynamics.
[4] Fluid-structure interaction.

3 Pre-processing

3.1 Geometry and case description

The studied airfoil is a NACA 63A415, used in the Embraer EMB 312 Tucano, because of its good behaviour in subsonic flight.

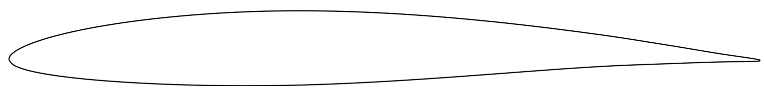

Figure 1: NACA 63A415.

Several 2D cases have been calculated [2] with different flow velocity in order to study the moment when flutter takes place.

3.2 Domain and Mesh

Regarding the computational domain, widely-accepted dimensions [3] have been used, which can be seen in Figure 2.

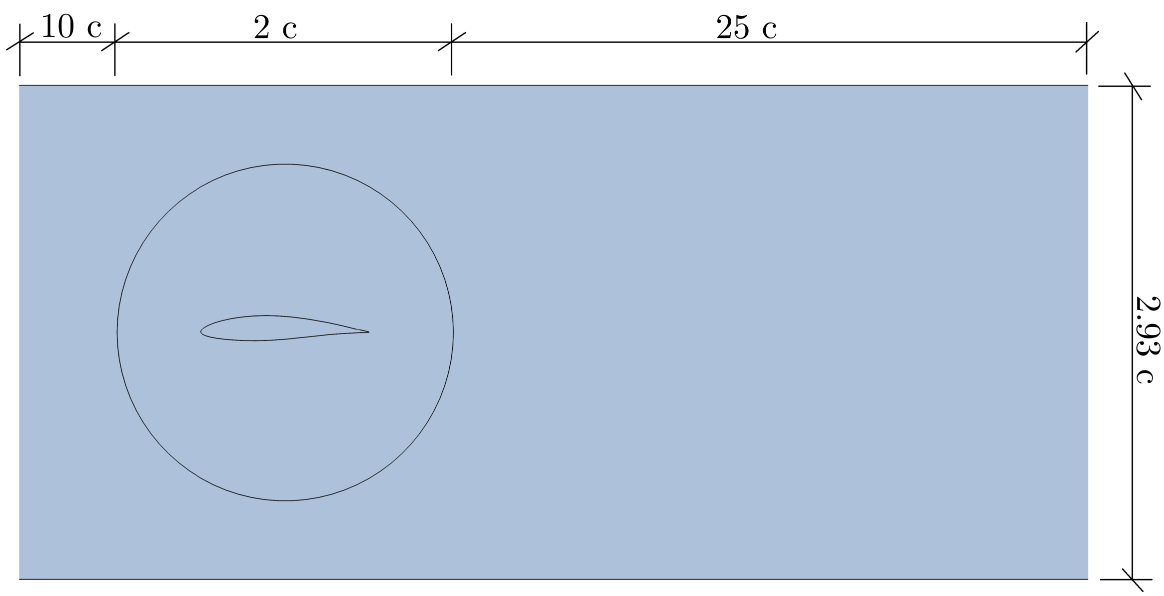

Figure 2: Calculation domain (dimensions are not scaled).

- 10 chords upstream.
- 25 chords downstream.

- 2.93 chords in perpendicular directions. This distance is the width of the wind tunnel used in the experimental study, so this way it is possible to reproduce these conditions where wall effects affect the results.

The chosen mesh is polyhedral with extruded polygonal shapes. Apart from the regular mesh, an overset mesh has been made near the airfoil. This mesh will be able to move, interpolating the results between the two meshes. The elements in the area near the body are smaller.

A mesh independence study has been carried out. It was achieved with 0.5 M of elements. However, the differences with a mesh with 0.1 M of elements were smaller than 1%. As a result, in order to reduce computational cost, this was the mesh which was used (see Figure 3).

It has been considered a boundary layer of 6 elements with a total thickness of 2 mm. This way, the values of $y+$ are between 0 and 4 (below the maximum recommended value of 5 in this kind of problems [4]).

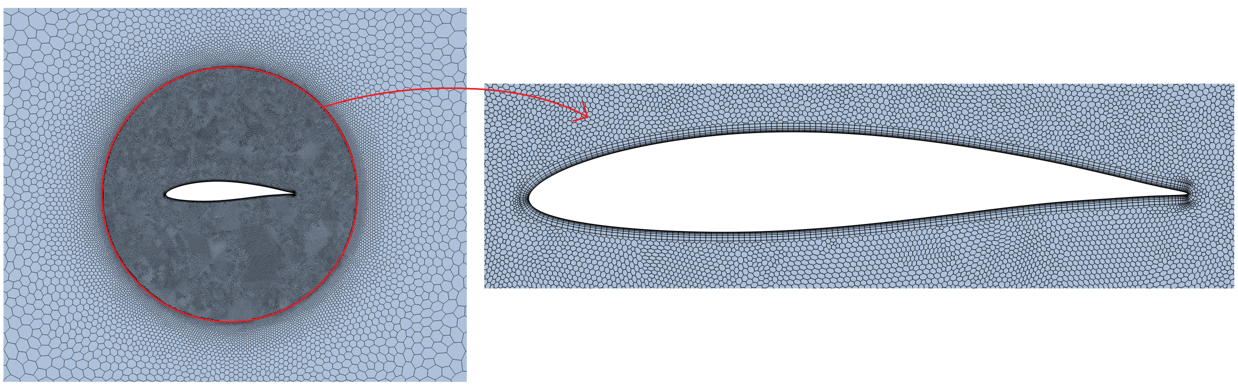

Figure 3: Detail of the mesh used for the calculation.

3.3 Turbulence modelling, solver and movement analysis

Turbulence modelling: the study is unsteady, so an URANS (Unsteady Reynolds-averaged Navier–Stokes) model has been used, in particular, a $SST\ k-\omega$ model [5, 6], which solves some of the problems of the regular $k-\omega$ model regarding the influence of ω (specific rate of dissipation) far from the object in the calculations of the boundary layer.

Solver: segregated solver, which is stable and quicker for subsonic flow. A second-order upwind scheme has been used. The equations are solved implicitly with a time step of 10^{-5} s.

Movement analysis: the airfoil is modelled as body with 2 available degrees of freedom (vertical displacement and pitch), restricted by linear springs.

3.4 Boundary conditions

The boundary conditions which have been used are the following ones:

- *inlet*: desired flow velocity.
- *outlet*: sea level pressure (according to the wind tunnel location).
- *wall*: wall condition in the airfoil.
- *overset*: in the area near the airfoil.
- *simmetry*: simmetry condition in the rest of surfaces.

4 CFD results

All the cases (with different incident flow velocities) are represented in Figure 5 (evolution of angle of attack) and 6 (evolution of vertical displacement).

- With 2 m/s, the movement is minimum (see Figure 4).
- With 9 m/s, the oscillating movement dampens, although this happens rather slowly.
- With 9.5 m/s, the amplitude of the movement increases after the first cycles. As a result, this will be considered as the flutter velocity (see Table 1).
- With 10 m/s, the movement amplitude increases much faster.
- With 11 m/s, the amplitude always increases, reaching more than 20º of angle of attack in less than 1 second of calculation.

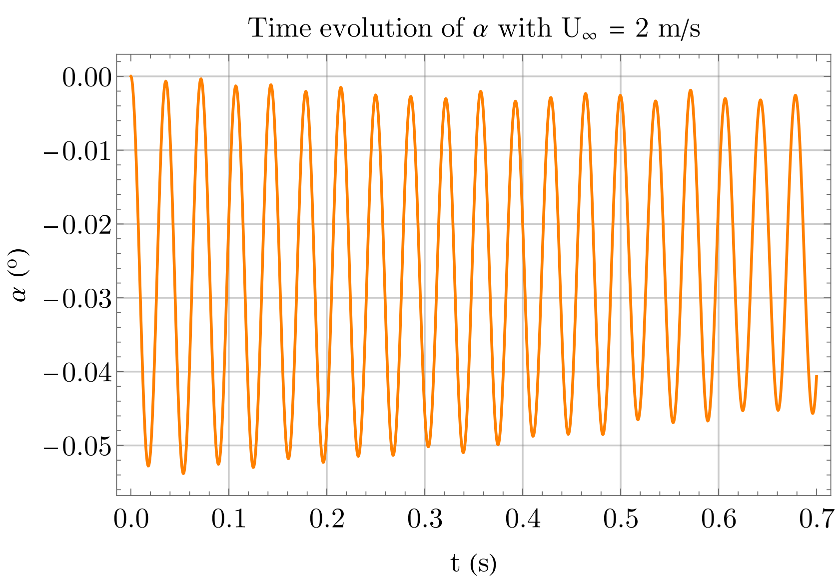

Figure 4: Time evolution of the angle of attack with $U_\infty = 2$ m/s.

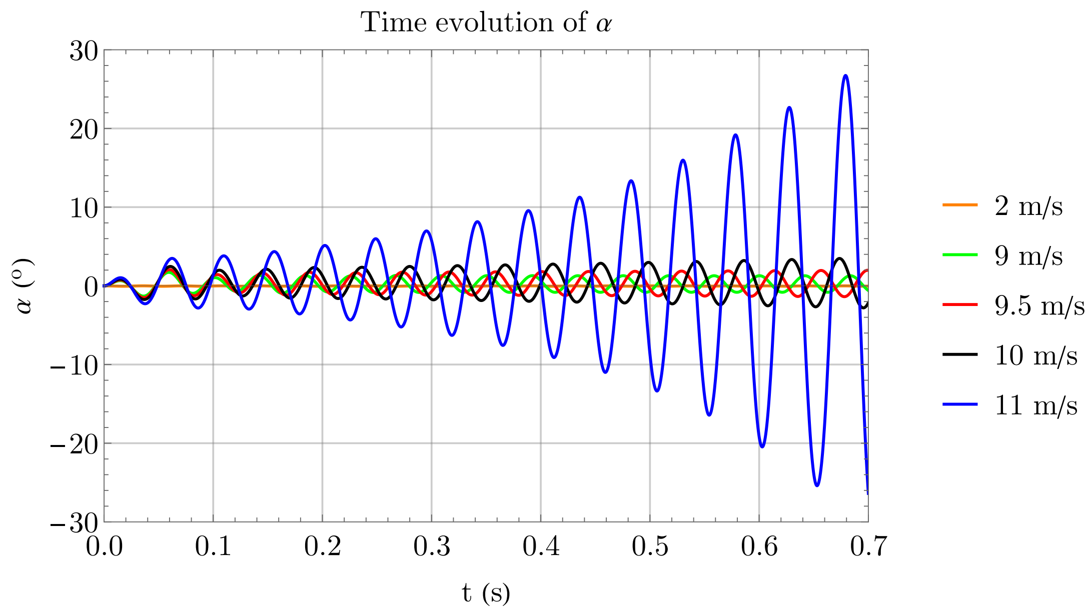

Figure 5: Evolution of the angle of attack for different velocity values of the incident flow.

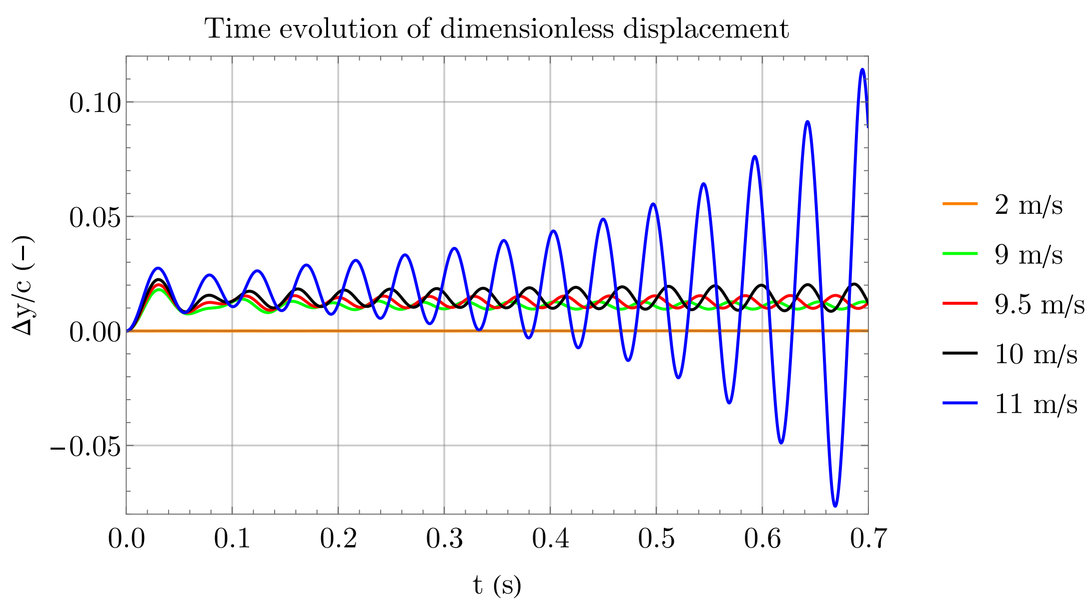

Figure 6: Evolution of the dimensionless vertical displacement for different velocity values of the incident flow.

Flutter velocity (m/s)	9.5
Flutter frequency (Hz)	23.5

Table 1: Numerical results.

For $U_\infty = 11$ m/s the response turns unstable much faster, which can be seen in Figure 7.

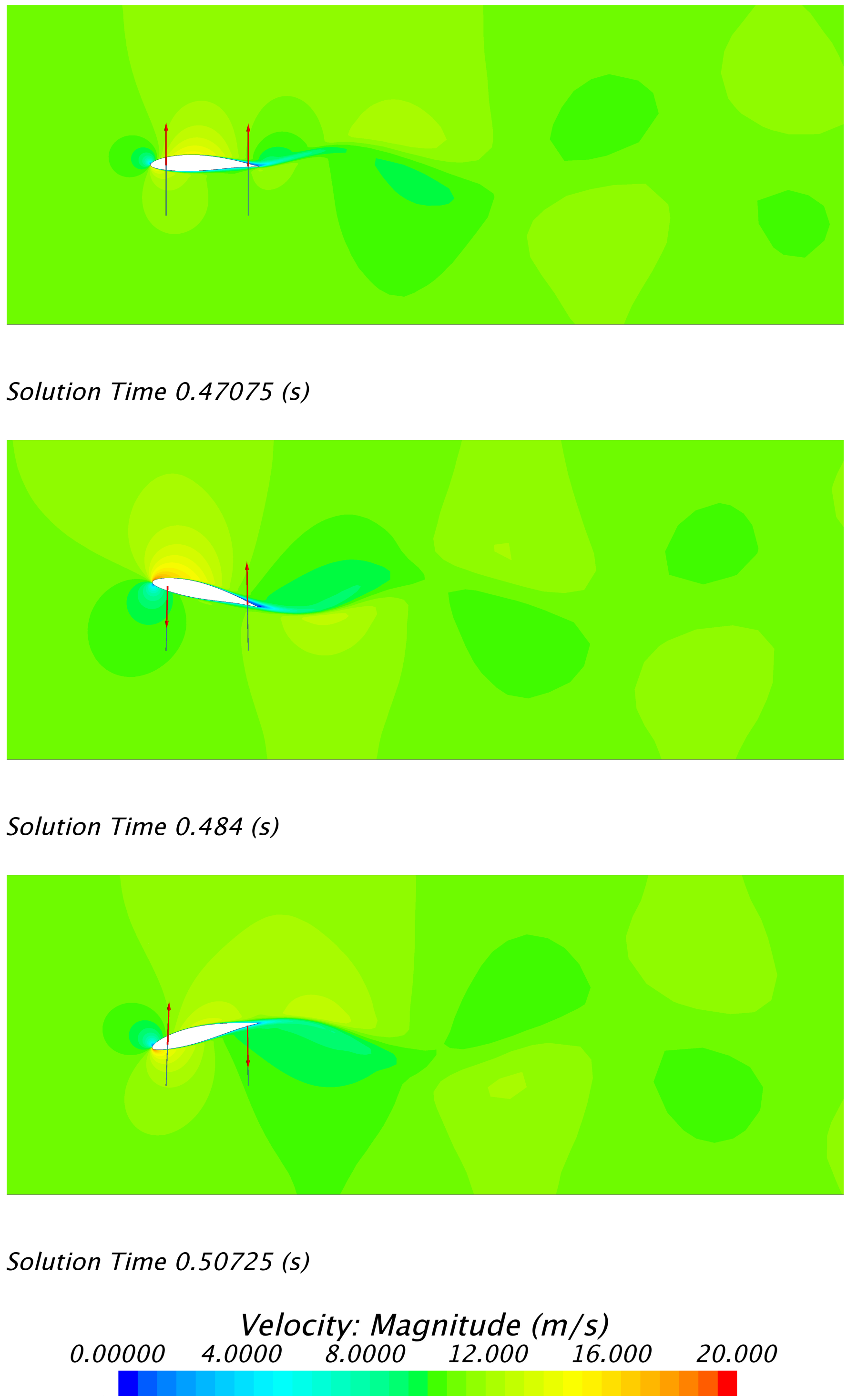

Figure 7: Flutter evolution with $U_\infty = 11$ m/s.

5 Wind tunnel testing

The airfoil has been tested in a wind tunnel so as to determine flutter velocity and frequency. The wind tunnel used in this experimental study is the closed-circuit wind tunnel of the UPV. The measurement section has 44 cm of side, with optical access and it is possible to test up to 80 m/s of flow velocity.

The wing has been made with Expanded Polystyrene (EPS) and cut with a hot wire machine. It has also been covered with plastic in order to reduce surface roughness.

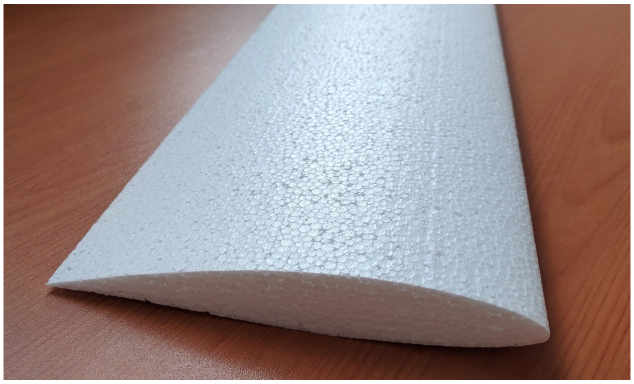

Figure 8: EPS wing.

The experiment is represented in Figure 9 where linear springs have been used in order to allow the model the desired degrees of freedom (horizontal displacement is not restricted but it is insignificant), as well as representing the real wing rigidity.

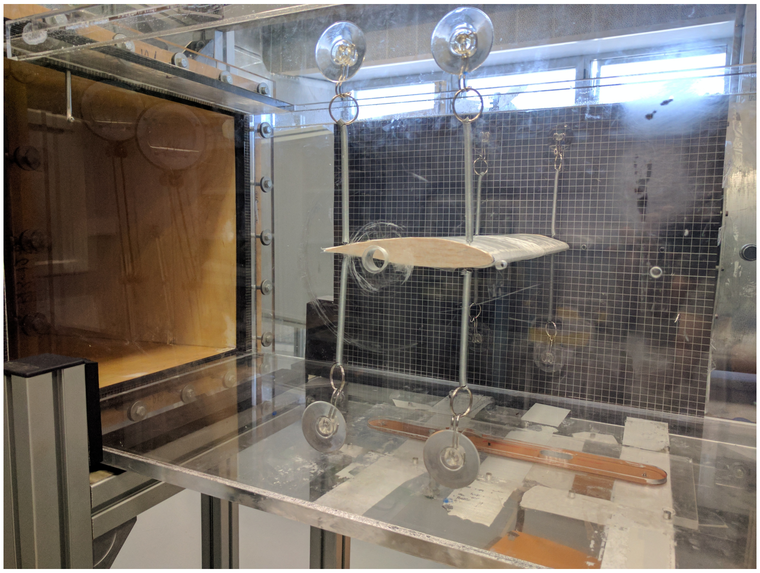

Figure 9: Wind tunnel set up.

The experiment consists on gradually increasing the flow velocity until flutter can be observed. The phenomenon is filmed using 2000 fps. That way, the frequency at which it takes place can be calculated.

6 Wind tunnel results

In Figure 10 it can be seen an example where flutter has already started.

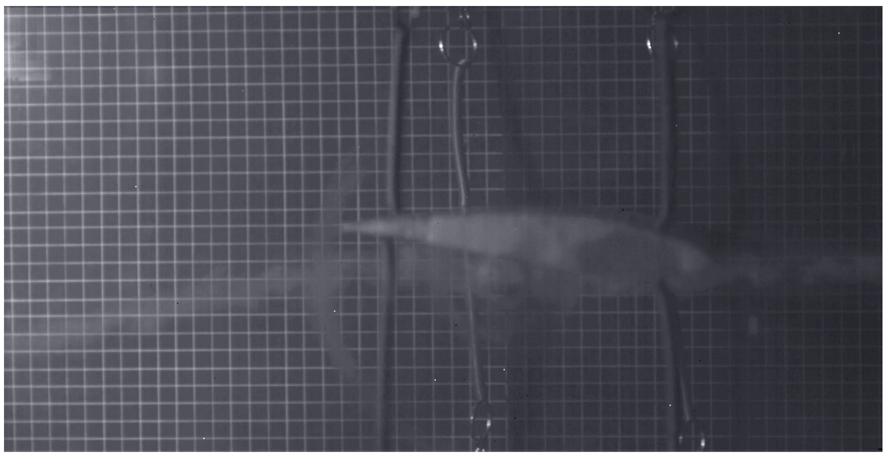

Figure 10: Airfoil's flutter with $U_\infty = 11$ m/s.

From the different experiments, the results obtained are the ones which follow.

Flutter velocity (m/s)	10.690
Flutter frequency (Hz)	24.691

Table 2: Experimental results.

Comparing these experimental results with the numerical ones in Table 1, it can be observed that they are quite similar.

7 Conclusions

In this study, CFD with FSI studies have been proved as a useful tool in order to study and calculate the phenomenon of flutter. The results obtained in the wind tunnel are similar to the ones obtained with CFD, validating the method.

With the available computational resources, in approximately 8 hours, it is possible to calculate 1 second of simulation. As a result, this method could be used in the industry and research to the extent possible so as to avoid long and expensive experimental studies.

Furthermore, this way, different and complex wing configurations could be calculated in short periods of time, even with 3D calculations, getting much more detail in the results than with classical methods of flutter calculation.

Nomenclature

α	—	Angle of attack (º).
Δy	—	Vertical displacement (m).
c	—	Airfoil chord (m).
t	—	Time (s).
U_∞	—	Flow velocity (m/s).
y^+	—	Dimensionless wall distance (-).

References

[1] Raymond L. Bisplinghoff and Holt Ashley. *Principles of Aeroelasticity*. Dover Phoenix Editions, 2002.

[2] *STAR-CCM+ 12.02.010 User Guide*. 2017.

[3] J.D. Anderson Jr. *Computational Fluid Dynamics: The Basics with Applications*. McGraw-Hill, 1995.

[4] Salim .M. Salim and S.C. Cheah. Wall y+ strategy for dealing with wall-bounded turbulent flows. *Proceedings of the International MultiConference of Engineers and Computer Scientists*, 2009.

[5] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA journal*, 32(8):1598–1605, 1994.

[6] F. R. Menter, M. Kuntz, and R. Langtry. Ten years of industrial experience with the sst turbulence model. *Turbulence, heat and mass transfer*, (4):625–632, 2003.