NACA 4415 Polar Curve

Realization of the polar curve of the airfoil by CFD simulation

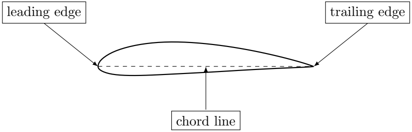
Andrea Marchegiani
andrea.marchegiani@studenti.unitus.it

UNIVERSITÀ DEGLI STUDI DELLA TUSCIA

DIPARTIMENTO DI ECONOMIA, INGEGNERIA, SOCIETÀ E IMPRESA

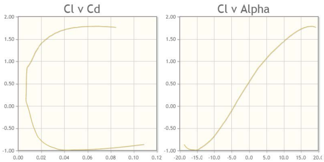
Master's Degree in Mechanical Engineering
Numerical Thermo Fluid Dynamics

Table of Contents



Introduction

The aim of this project is to evaluate the polar curve of the NACA 4415 airfoil (figure 2) by the validation of a CFD model.



This particular representation permit to individuate, by the tangent curve for the origin, the best attack angle to optimize the working airfoil.

In this point, the maximum $\dfrac{C_L}{C_D}$ permits to have the maximum lift over the minimum drag.

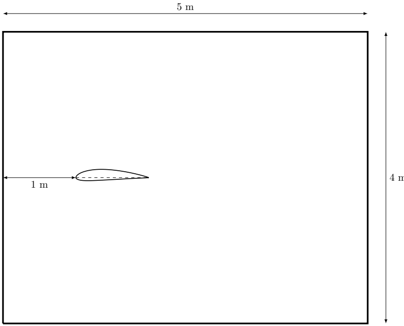
Materials & Methods: Software & Hardware

Software

- ANSYS 2023 R2 Student version;
- Geometry: SpaceClaim
- Calculus: Fluent

Hardware

- CPU:
  AMD FX 8150 8-core processor
- GPU:
  ASUS NVidia GeForce GTX 1660 Ti
- RAM: 16 GB



Initial Data:

- Airfoil geometry;
- $Re = 1 \times 10^6$, $\rho = 1.225\,\mathrm{kg\,m^{-3}}$,
  $\mu = 1.7894 \times 10^{-5}\,\mathrm{kg\,m^{-1}\,s^{-1}}$, $L = 1\,\mathrm{m}$, $\Rightarrow$ $u = 14.607\,\mathrm{m\,s^{-1}}$

Materials & Methods: Models & Scheme

Brief Analysis - Hypothetical value of mesh size for testing model and method (default setting for free steam mesh)											
Edge sizing	Mesh size	Max Skewness	%Skw	State	Model	y^+	Method	Iterations	C_L	Δ	err%
0.00012	0.3	0.75	0.15	Steady	Viscous $k\omega$ SST	5	SIMPLE	1068	0.39835492	0.015300708	3.698899994
							SIMPLEC	1737	0.36678985	0.046865778	11.32966043
							PISO	1055	0.39921114	0.014444488	3.491911393
							Coupled	79	0.34955502	0.064100608	15.49612858
0.00381		0.7	0.04		Viscous $k\varepsilon$ sdt	165	SIMPLE	439	0.34748226	0.066173368	15.99721206
							SIMPLEC	610	0.35107735	0.062578278	15.1281099
							PISO	438	0.34744313	0.066212498	16.00667162
							Coupled	70	0.35038373	0.063271898	15.29579044
0.00265		0.62	0.11		Spallart - Allmaras VB	115	SIMPLE	745	0.3298986	0.083757028	20.24800881
							SIMPLEC	854	0.32843196	0.085223668	20.6025646
							PISO	565	0.3297579	0.083897728	20.28202261
							Coupled	65	0.32774016	0.085915468	20.76980517

By evaluating the flow impacting the airfoil at 0°, we can choose the best model/method coupling.

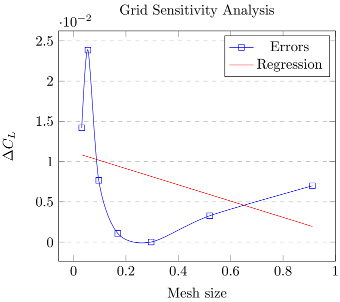
Viscous kω SST model with the PISO method seems to be the best coupling at all.

Knowing that PISO make an extra pressure correction step, for the steady state simulation is maybe quite a waste, by the way, knowing also that in this simulation will be a transient case from -8° to -16° and from 8° to 16 °, adding ab extra correction step lead maybe to a more accurate results.
Nevertheless, its preferable working with the same method for all the cases.

Grid Sensitivity Analysis

An analysis regarding the sensitivity of the grid will now be provided, starting from a coarse mesh using the methods and parameters just now studied.

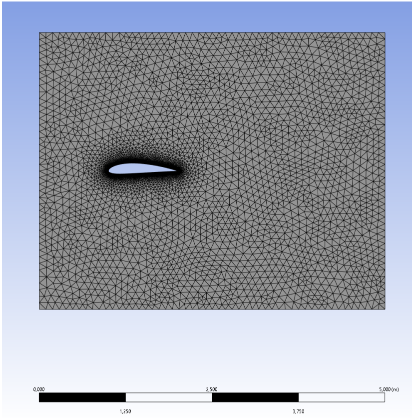
Mesh Sensitivity Analysis RF = 0.57					
Mesh size	C_L	Δ	Iter	Max Skw	%Skw
1.6	0.392276		1165	0.75	0.16
0.912	0.399265	0.006989	1060	0.748	0.18
0.51984	0.40253	0.003265	1019	0.733	0.25
0.296309	0.402525	4.62E-06	1052	0.771	0.1
0.168896	0.401451	0.001074	1048	0.76	0.13
0.096271	0.393783	0.007668	1163	0.772	0.1
0.054874	0.369932	0.023851	1762	0.739	0.21
0.031278	0.384137	0.014204	1428	0.751	0.14



These are so the chosen parameters:

Model	Viscous $k\omega$ SST
Method	PISO
Edge Size	0.00012
Mesh size	0.09627

That leads to this mesh:



With these general conditions, on the boundaries:

- **Inlet**: velocity inlet type;
- **Outlet**: pressure outlet type;
- **Symmetry**: applied on the top and the bottom of the bounding box;
- **Wall**: on the airfoil edges, no slip conditions are applied.

And on the method:

- **Skewness Correction** and **Neighbor Correction** equals to 1;
- **Gradient**: leas squares cell cased
- **Pressure**: second order
- **Momentum**: second order upwind
- **Turbolent Kinetic Energy**: second order upwind
- **Specific Dissipation Rate**: second order upwind

Under Relaxation Factors

URF choice 0°				
Iterations	Controls	C_L	ΔC_L	err %
1163	Default	0.39378324	0.019872388	4.804089841
912	0.4 P	0.41253355	0.001122078	0.271258971
647	0.5 P	0.42725715	0.013601522	3.288126905
902	0.4 P - 0.5 M	0.41844258	0.004786952	1.157231203
925	0.4 P - 0.5 M - 0.6 TKE, SDR	0.41887024	0.005214612	1.260616718

URF choice 2°				
Iterations	Controls	C_L	ΔC_L	err %
1046	Default	0.603911	0.020460743	3.27701276
974	0.4 P	0.612075	0.012296813	1.969469687
672	0.5 P	0.631829	0.007457117	1.194339207
979	0.4 P - 0.5 M	0.619449	0.004922643	0.788415353
929	0.4 P - 0.5 M - 0.6 TKE, SDR	0.619589	0.004782993	0.766048872

URF choice -2°				
Iterations	Controls	C_L	ΔC_L	err %
1125	Default	0.19121338	0.034326	15.21934294
901	0.4 P	0.20744404	0.018095	8.022953129
621	0.5 P	0.22352259	0.002016	0.894006224
815	0.4 P - 0.5 M	0.21978717	0.005752	2.550225898
823	0.4 P - 0.5 M - 0.6 TKE, SDR	0.21987279	0.005666	2.512263493

After providing the grid sensitivity analysis, now will be evaluated the right choice of the URFs by their combination, for providing the most accurate results.

Are used -2°, 0°, 2° cases, and their experimental data.

With this analysis make from the study of the behavior of 3 point, emerges that the best URFs choice are the 0.4 on Pressure and 0.5 on Momentum due to a combination of less iterations and minor error from the experimental data:

- -2°: 815 iteration, 2.55% C_L error;
- 0°: 902 iteration, 1.15% C_L error;
- 2°: 979 iteration, 0.78% C_L error.

Transient Analysis

The transient analysis is provided by the choose of the *Courant* number. In the vast majority of cases, it can be assumed equal to 10.
By the C number we may know the necessary time step for the transient simulation.

$$C = \frac{u \cdot \Delta t}{\Delta x}$$

In which u is the free flow velocity, Δx is the minimum value of mesh elements and the Δt is the time step size.

$$\Delta t = \frac{C \cdot \Delta s}{u} = 8 \times 10^{-5}\,\text{s}$$

For evaluating the best number of time steps are token the $-10°$ and $10°$ cases.

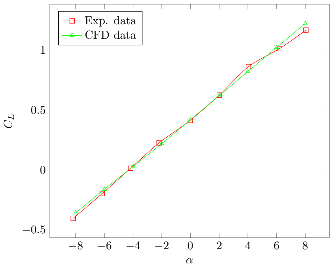
$10°$						
N. time step	It x time step	Δt	Flow time [s]	C_L	ΔC_L	err %
1500	20	8.00E-05	1.20E-01	1.1499	0.110258	8.749544
2500			2.00E-01	1.24766	0.012498	0.991772
3750			3.00E-01	1.310601	0.050443	4.002872
$-10°$						
1500	20	8.00E-05	1.20E-01	-0.53838	0.063893	-10.6088
2500			2.00E-01	-0.58119	0.021081	-3.50021
3750			3.00E-01	-0.58907	0.0132	-2.19179

For the $10°$ case, 2500 time steps lead to a more accurate results, while for the $-10°$ case, same time steps value lead to a not-so-bad results with less computational capacity and time.

Results: C_L Curve

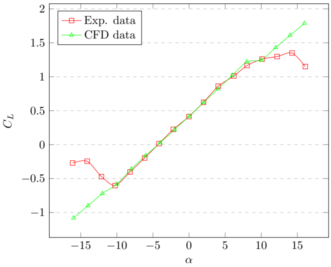
C_L vs α			
Exp. data		CFD data	
α	C_L	α	C_L
-8.17377	-0.40285	-8	-0.35806
-6.14642	-0.1959	-6	-0.16057
-4.16734	0.01482	-4	0.027939
-2.18825	0.225539	-2	0.219787
-0.01609	0.413656	0	0.418443
2.011263	0.624372	2	0.619449
4.038616	0.861443	4	0.821015
6.21078	1.011909	6	1.021107
8.045052	1.166162	8	1.220264

C_L vs α



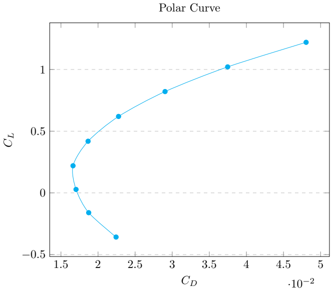
C_L vs α			
Exp. data		CFD data	
α	C_L	α	C_L
-16.1384	-0.27057	-16	-1.0802
-14.111	-0.24435	-14	-0.89894
-12.1319	-0.47037	-12	-0.72026
-10.2494	-0.60227	-10	-0.58119
-8.17377	-0.40285	-8	-0.35806
-6.14642	-0.1959	-6	-0.16057
-4.16734	0.01482	-4	0.027939
-2.18825	0.225539	-2	0.219787
-0.01609	0.413656	0	0.418443
2.011263	0.624372	2	0.619449
4.038616	0.861443	4	0.821015
6.21078	1.011909	6	1.021107
8.045052	1.166162	8	1.220264
10.12068	1.260158	10	1.24766
12.1963	1.297678	12	1.429237
14.27192	1.350259	14	1.61119
16.10619	1.150596	16	1.788569

C_L vs α

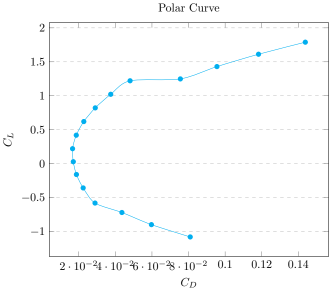


Results: Polar Curve

C_L *vs* C_D	
CFD data	
C_L	C_D
-0.35806	0.022427
-0.16057	0.018739
0.027939	0.01703
0.219787	0.016623
0.418443	0.018653
0.619449	0.022761
0.821015	0.029031
1.021107	0.037473
1.220264	0.048051



C_L *vs* C_D	
CFD data	
C_L	C_D
-1.0802	0.080922
-0.89894	0.059716
-0.72026	0.043587
-0.58119	0.028905
-0.35806	0.022427
-0.16057	0.018739
0.027939	0.01703
0.219787	0.016623
0.418443	0.018653
0.619449	0.022761
0.821015	0.029031
1.021107	0.037473
1.220264	0.048051
1.24766	0.075529
1.429237	0.095512
1.61119	0.11824
1.788569	0.143804



Conclusions

We can see that the CFD model studied provides an excellent approximation for the linear case, from 8° to -8°.
By the way, at the externals region of this point, from -10° and 10°, the model is absolutely not suitable: the detachment of the fluid vein and the incidences of vorticity do not allow a correct evaluation of this model.