International Core Journal of Engineering Volume 7 Issue 1, 2021
ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

Airfoil Shape Optimization for Low Reynolds Number Flow

Yongsong Huang[1], Chenggeng Xie[2], Xiaoqiang Gao[3]

[1]College of Engineering & Computer Science, Syracuse University, Syracuse, NY, 13244, USA;

[2]School of Aerospace Engineering, University of Nottingham Ningbo China, Ningbo, Zhejiang 315000, China;

[3]Leeds Joint School, Southwest Jiaotong University, Chengdu, Sichuan 611756, China.

Abstract

When air passes through the wing of an aircraft, lift and drag will be produced due to the pressure difference between the upper surface and lower surface. Lift-to-drag ratio, an important term that shows the aerodynamic efficiency, mainly depends on the shape of the airfoil. To increase the lift-to-drag ratio, it is necessary to optimize the airfoil by changing its camber, camber location or the thickness. Javafoil and Ansys Fluent are used to do the modeling and analysis processes to find the maximum value of the lift-to-drag ratio of an NACA airfoil at certain angle of attack for low Reynolds Number flow. By converge the results, we can see the effects of these essential parameters on the performance of an airfoil, which may contribute to the further wing design optimization.

Keywords

Airfoil Optimization; Lift-to-Drag Ratio; Javafoil; Ansys Fluent.

1. Introduction

If an object that moves through a kind of fluid, a pressure field will be induced in its vicinity. The pressure field changes the pressure at the surface and produce a resultant reaction pressure force, R, acting on the object. Lift, L, is defined as the component of this force which is normal to the trajectory or the flight path. Drag, D, is defined as the component of this force tangent to the trajectory [1]. The purpose of this project is to focus specially on the important geometric shape used for lifting surfaces: the airfoil. What sets the airfoil shape apart from other geometry is that its resultant force approaches being normal to the tangent to the trajectory. This results in a lift force component substantially larger than the drag component. When it comes to aircrafts, we are mostly interested in bodies whose geometry results in a lift force that is substantially larger than the drag. Airfoils are examples of such bodies. Date back about 800 years, human found that the flat surface in the wind would produce a sideway force which was called lift now. In 1800's, some people suspected if the shape with curvature could be more closely resembled to the bird wings, it would gain more lift or be more efficient [2]. During this time, the NACA collected and developed many kinds of airfoils and created the 'families' airfoils. Some of them were still used today and had a meaningful impact on the future airfoils [3].

Airfoil is the cross-section shape of the aircraft wing or the blade of a propeller. To understand how airfoil works, some basic concepts need to be clarified. Chord line is a straight line that connects the leading edge and the trailing edge. The length of the chord line c is related to the wing area S, where $S = b \cdot c$ (b is the wingspan). Aspect ratio, $AR = \frac{b}{c} = \frac{b^2}{S}$, presents the dimension and size of the wing. Camber line also connects the leading edge and trailing edge, however, every point on the line has equal distance to the upper surface and the lower surface of the wing. Camber line shows the curvature

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

of the wing. For general aircrafts that create upwards lift at zero angle of attack, the camber line is above the chord line and the distance between the two lines is called camber.

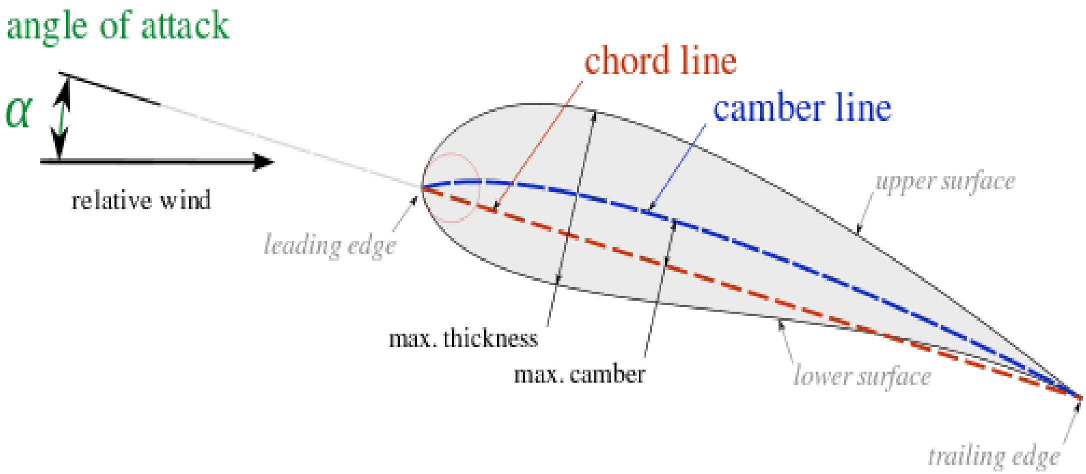

Figure 1. Airfoil terminology [4]

When an aircraft travels in the air, there are four main forces acting on it which are Lift, Weight, Thrust, and Drag. Weight is related to the mass of the aircraft and thrust is produced by the engines. During cruise, the air velocity at the upper surface of the wing is larger than the velocity at the bottom surface, caused by their difference in profile (airflow needs higher velocity to travel through the upper longer curvature for the same time). This velocity variance leads to a higher pressure at the lower surface and a lower pressure at the upper surface (Bernoulli Equation and Euler's Equations can both verify it), hence generate upwards lift on the wing area.

Lift equation: $L = \bar{q}SC_L$, where $\bar{q}$ is the dynamic pressure $\bar{q} = \frac{1}{2}\rho v^2$, S is the wing area, C_L is the lift coefficient. For reasonable range of angle of attack (no stall), the lift increase as the angle of attack increases. This relationship can be expressed in a linear function. The airflow in this stage is attached to the surface of wing section.

$$C_L = C_{L_0} + C_{L_\alpha} \cdot \alpha$$

C_{L_0} is zero angle of attack lift coefficient, C_{L_α} is the slope. These values for different airfoils are determined from wind tunnel test or numerical aerodynamics calculations.

Drag is the resistance to the movement of an aircraft. There are two types of drag. Parasite drag is caused by the physical shape of the aircraft and becomes higher when airspeed increases. Induced drag can be regarded as a byproduct of lift and varies with the angle of attack

$$C_D = C_{D_0} + kC_L{}^2$$

C_{D_0} is zero lift coefficient or parasite drag coefficient, $kC_L{}^2$ is associated with the induced drag, k is the induced drag coefficient $k = \frac{1}{\pi e AR}$. Therefore, large aspect ratio means small induced drag while small aspect ratio means large induced drag.

Lift-to-drag ratio is of tremendous significance because it measures the aerodynamic quality of an airfoil. Higher $\frac{L}{D}$ leads to better maneuverability and endurance.

Simplify: $\frac{L}{D} = \frac{C_L}{C_D}$

ISSN: 2414-1895
DOI: 10.6919/ICJE.202101_7(1).0046

Maximum lift-to-drag ratio usually happens when the drag is at the minimum value where the parasite drag is equal to the induced drag. At $C_{D_{min}}$,

$$C_L = \sqrt{\frac{C_{D_0}}{k}}$$

Therefore, to maximize lift-to-drag ratio, further research needs to be done on the profile of the airfoil. In the work, our objective is to maximize the lift-to-drag ratio of an aircraft and the approach is to optimize the shape and profile of the airfoil. We start with a symmetric airfoil NACA 0012 and gradually changed its profile in a software called Javafoil by changing its camber, camber location and thickness. Later we roughly compare the models at different angle of attack and put the good ones into Ansys Fluent to do further analysis. By doing so, we can obtain their lift-to-drag ratio for low Reynolds Number flow and find the maximum value by comparison.

2. Problem Statement

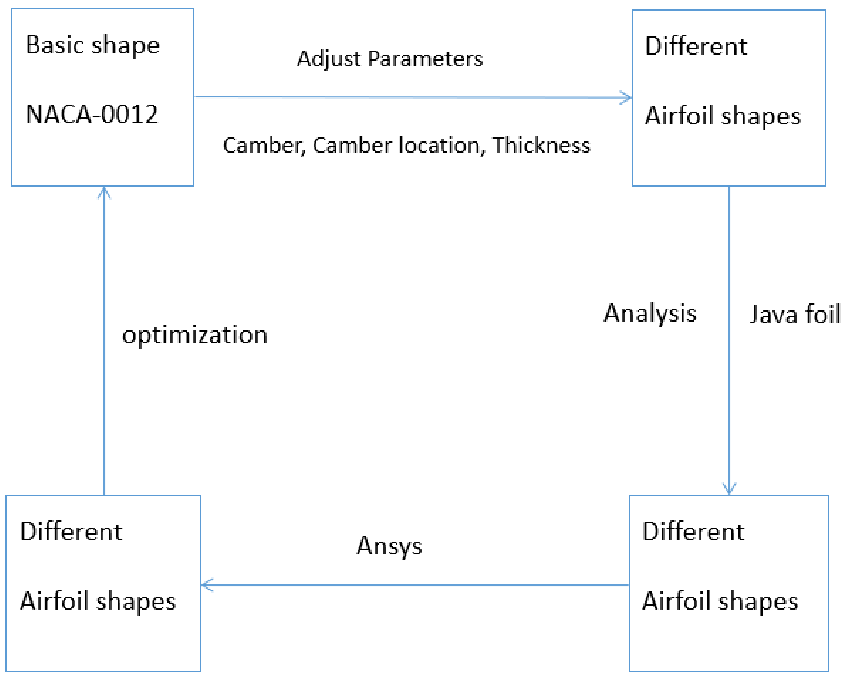

Figure 2. Research Schematic

The ratio of coefficient of lift to drag of an airfoil is expected to be as high as possible, which is of great significance to the design of the aircraft [5]. Currently, several essential parameters of the airfoil, including camber, camber location, and thickness need to be adjusted in order to get the maximum ratio of coefficient of lift and drag as well as the most optimal shape of the airfoil. The high the ratio of the lift coefficient to the drag coefficient means that the aircraft is more efficient and energy-saving during flight. To optimize the shape of the airfoil, normally genetic algorithm and parameterization methods or other complex methods are used. Here in this case, the optimization is started with a symmetric airfoil NACA0012, and by changing the thickness, camber and camber location, the impact of these parameters on the lift-to-drag ratio of the airfoil at different angle of attack is observed. For convenience, a software called "Javafoil" is used for generating the profile of the airfoil and doing simple simulations on the lift-to-drag ratio. After that, three top airfoils can be selected to have larger lift-to-drag values and imported into Ansys Fluent. Ansys Fluent can provide better analysis and simulations, which are more accurate and reliable. The results may be compared with the former ones, hence having a cross-validation on the work done by Javafoil..

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

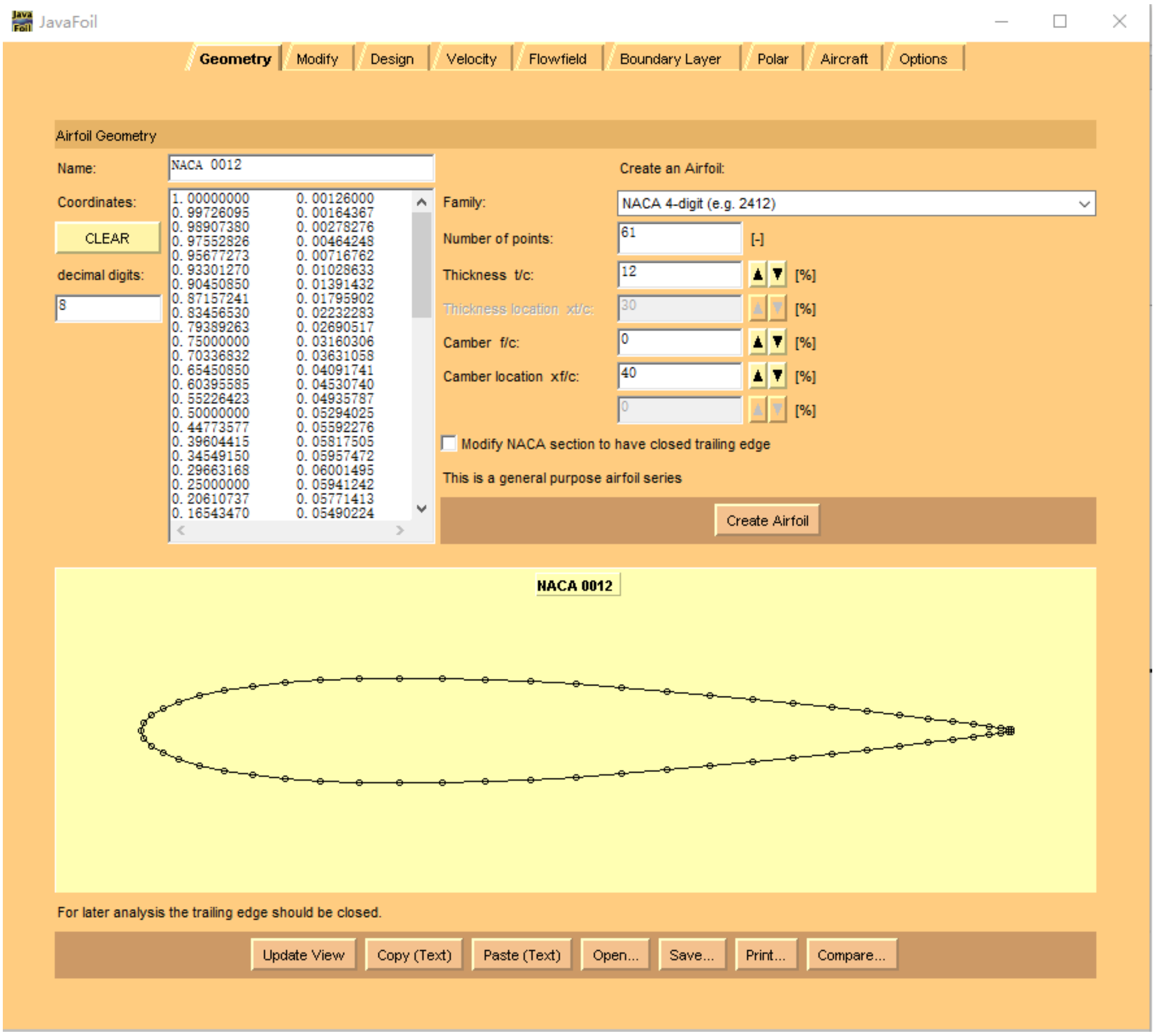

Figure 3. Javafoil

For clarification, Javafoil is a simple but powerful software which has a validation that can correctly calculate many parameters, indicating the test condition of the airfoil as well as the flow stays subsonic. It utilizes several traditional methods for airfoil analysis. The panel method contributes to the potential flow analysis, calculating the local inviscid flow velocity along the surface of an airfoil due to sets of coordinates. The boundary layer method focuses on solving differential equations on the purpose of finding different boundary layer parameters. Once the software examines the airfoil model, it will calculate the lift and pitching moment based on the distribution of the velocity on the surface. The data of flows at the boundary layer are integrated to give the friction drag, and the accurate total drag can be obtained using the "Squire-Young" approximation method. For different angle of attack, the above-mentioned steps will be repeated for one fixed Reynolds number ($Re = 40$) and give different results. Javafoil simulations do not model laminar separations and turbulent flow separation, which means that the outputs may be inaccurate when a large amount of flow separation occur, for example at stall angle of attack. However, this limitation is avoided as this project will not consider these extreme cases. Therefore, the theoretical background of Javafoil is evaluated to be reliable and convincing to run airfoil simulation programs and give accurate information.

3. Validation and Results Convergence

Before the analysis, two cases were simulated for validation. In the first validation case, the drag coefficient of square with an inclination of 45 degrees was built as the model in Ansys. The reference

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

values include $Depth = 3\ m$, $Length = 4.24\ m$, $Area = 4.24m \times 3m = 12.72\ m^2$. Air was the fluid with density of $1.225\ kg/m^3$ and viscosity of $1.7894e-5\ kg/m \cdot s$. The inlet velocity was set to $2.8\ m/s$, $pressure = 0\ Pa$, and Reynolds number was 600,000. Spalart-Allmaras model was selected since the Reynolds number was over 100,000. In this case, the reference value of the drag coefficient should be 1.55 [6]. According to the simulation in Ansys, the result of the drag coefficient was almost the same as the reference value, which shows in figure 5 and 6.

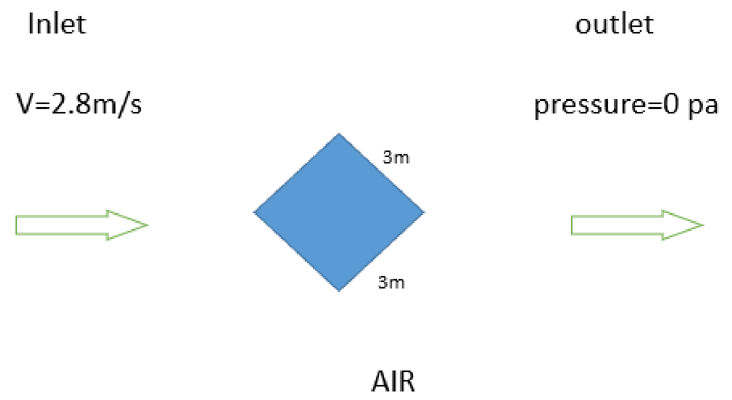

Figure 4. Schematic Diagram for case 1

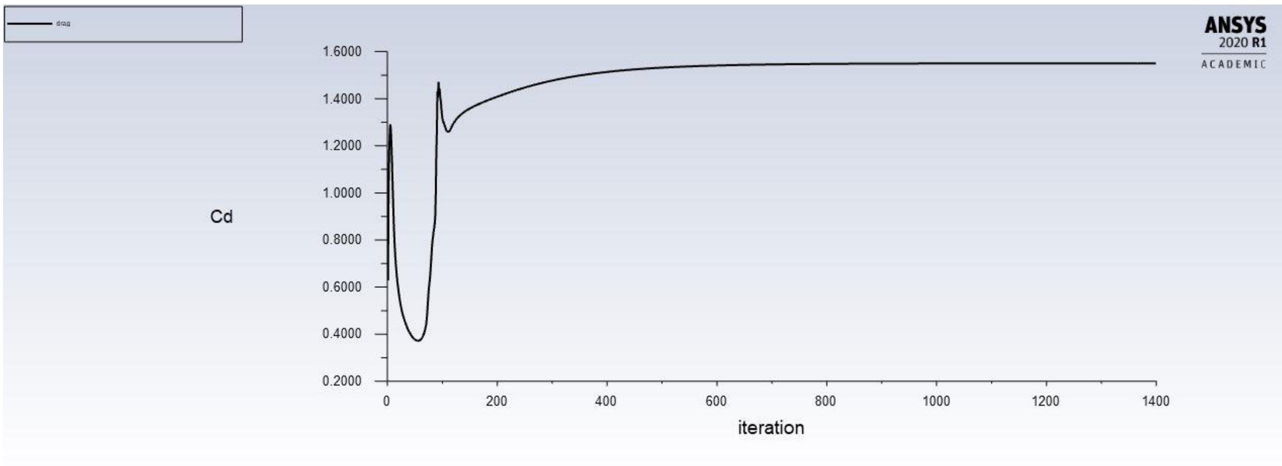

Figure 5. The convergence diagram of the drag coefficient and the iteration

Console

iter	continuity	x-velocity	y-velocity	nut	drag	time/iter	
1123	6.7843e-06	1.9728e-08	3.6307e-09	4.0524e-06	1.5499e+00	0:05:37	1377
1124	6.7375e-06	1.9594e-08	3.6059e-09	4.0255e-06	1.5499e+00	0:09:05	1376
1125	6.6912e-06	1.9461e-08	3.5813e-09	3.9988e-06	1.5499e+00	0:07:16	1375
1126	6.6451e-06	1.9329e-08	3.5567e-09	3.9722e-06	1.5499e+00	0:05:48	1374
1127	6.5994e-06	1.9198e-08	3.5324e-09	3.9458e-06	1.5499e+00	0:04:38	1373
1128	6.5541e-06	1.9067e-08	3.5083e-09	3.9196e-06	1.5499e+00	0:08:17	1372
1129	6.5092e-06	1.8938e-08	3.4843e-09	3.8935e-06	1.5499e+00	0:06:37	1371
1130	6.4646e-06	1.8809e-08	3.4607e-09	3.8676e-06	1.5499e+00	0:05:18	1370
1131	6.4203e-06	1.8681e-08	3.4373e-09	3.8419e-06	1.5500e+00	0:08:48	1369
1132	6.3764e-06	1.8554e-08	3.4139e-09	3.8164e-06	1.5500e+00	0:07:02	1368
1133	6.3327e-06	1.8428e-08	3.3905e-09	3.7910e-06	1.5500e+00	0:05:37	1367
iter	continuity	x-velocity	y-velocity	nut	drag	time/iter	
1134	6.2894e-06	1.8302e-08	3.3673e-09	3.7657e-06	1.5500e+00	0:04:30	1366
1135	6.2463e-06	1.8178e-08	3.3443e-09	3.7407e-06	1.5500e+00	0:08:09	1365

Figure 6. The convergence result calculated by Ansys

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

In the second validation case, a standard NACA 0012 airfoil with 12% thickness, 0 camber, and 40% camber location was simulated in the air fluid field. The inlet velocity was set to 40 m/s, and both the density and the viscosity of the fluid was set to 1. When the angle of attack was set to 10 degrees, the reference value of the lift coefficient from Javafoil was 0.87. In this case, the Viscous-Laminar model was used because the Reynolds number is relatively small. According to the simulation in Ansys, the result of the lift coefficient was 0.89 shown in Figure 8 & 9. The value has an error of 2% from the reference value given by Javafoil, which can be accepted.

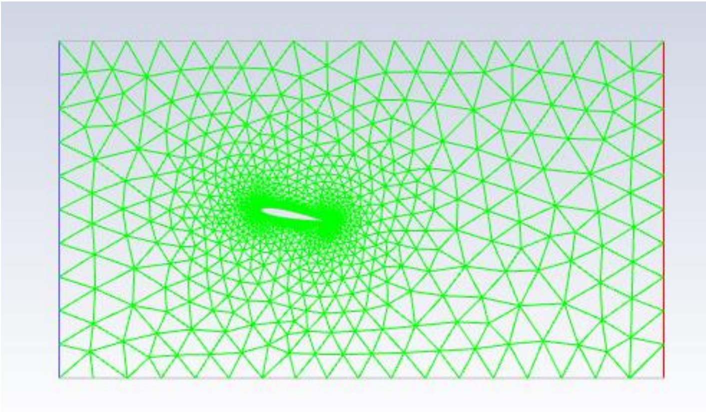

Figure 7. The simulation model for NACA0012 with the angle of attack of 10°

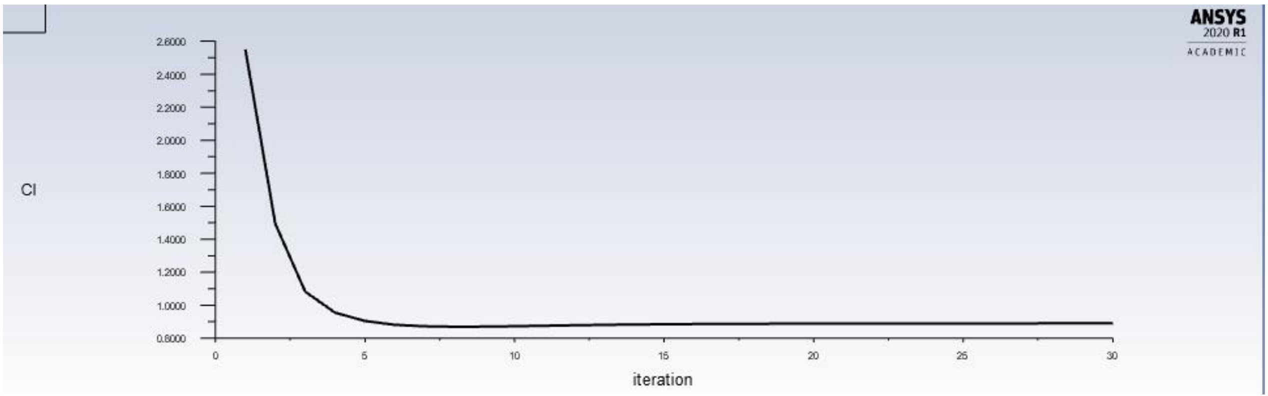

Figure 8. The convergence diagram of the lift coefficient and the iteration

iter	continuity	x-velocity	y-velocity	lift-c	time/iter	
23	4.8723e-03	1.2620e-05	7.1209e-06	8.8970e-01	0:00:19	77
24	3.8128e-03	9.6328e-06	5.4921e-06	8.8985e-01	0:00:15	76
25	3.0073e-03	7.4230e-06	4.2448e-06	8.8993e-01	0:00:12	75
26	2.3730e-03	5.6963e-06	3.2829e-06	8.9002e-01	0:00:09	74
27	1.8852e-03	4.5114e-06	2.5585e-06	8.9009e-01	0:00:07	73
28	1.5028e-03	3.5787e-06	2.0001e-06	8.9014e-01	0:00:06	72
29	1.2042e-03	2.8434e-06	1.5612e-06	8.9017e-01	0:00:04	71
! 30 solution is converged						
30	9.6786e-04	2.2689e-06	1.2177e-06	8.9019e-01	0:00:04	70

Figure 9. The convergence result calculated by Ansys

4. Mesh Convergence

Three meshes (M1, M2, M3. And M4) [7] are used to compute the fluid flow over the NACA 0012 airfoil at the angle of attack of 0° and Re=1. The details of the meshes are given in Table 1. The drag coefficients obtained for the NACA 0012 are also tabulated.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

Table 1. The drag coefficients of the NACA 0012 airfoil at α=0° and Re=1

Mesh	Nodes	Elements	Cd (NACA 0012)	Difference (%)
M1	950	1050	5.23	0
M2	1498	2269	4.86	7
M3	2080	3097	4.521	6
M4	3473	5427	4.516	0.1

Since the difference in the drag coefficients from M3 and M4 is less than 0.5% [7], mesh M3 shown in Figure 10 & 11 is used for further computations.

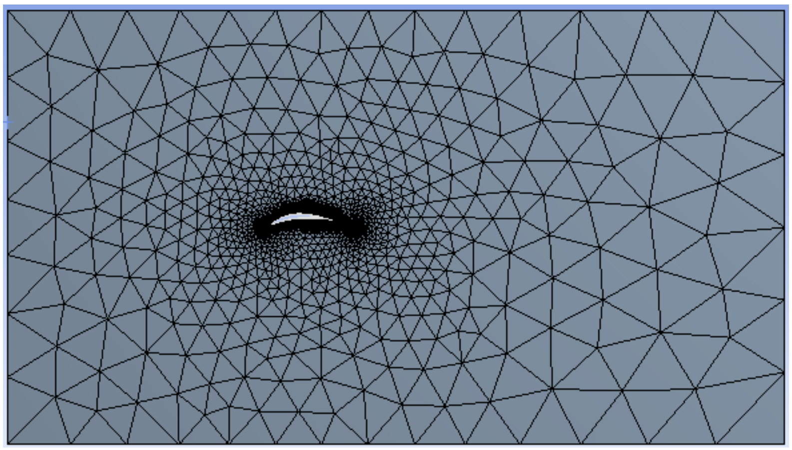

Figure 10. The convergence mesh in Ansys

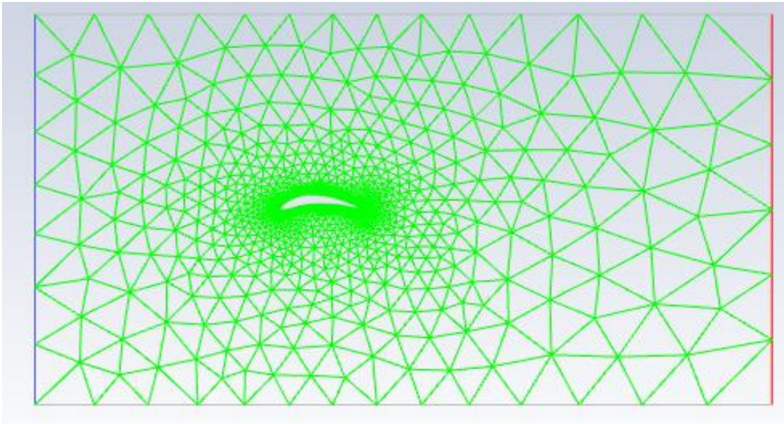

Figure 11. The convergence mesh in setup.

5. Optimization

After the convergence have been done, the first step is to change the value of the camber, thickness, and camber location of the basic airfoil. Only one single variable parameter is adjusted each time in Javafoil. The second step is to use the reference values of the ratio of lift coefficient to drag coefficient under different conditions given by Javafoil to plot the relations between the lift-to-drag ratio and different parameter variables. Optimal parameters can be obtained from the results [8]. In order to do a comparison and cross validation, the third step is to select three top optimal foils which had the highest lift-to-drag ratio to do the modeling and simulation in Ansys.

By changing different parameters of the basic NACA 0012 airfoil in Javafoil, specific coordinates of corresponding optimized airfoils can be obtained from Javafoil, and can be exported as a data file which was used to build the basic model of the airfoil by 3D curve in Ansys, which is shown in Figure 12.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

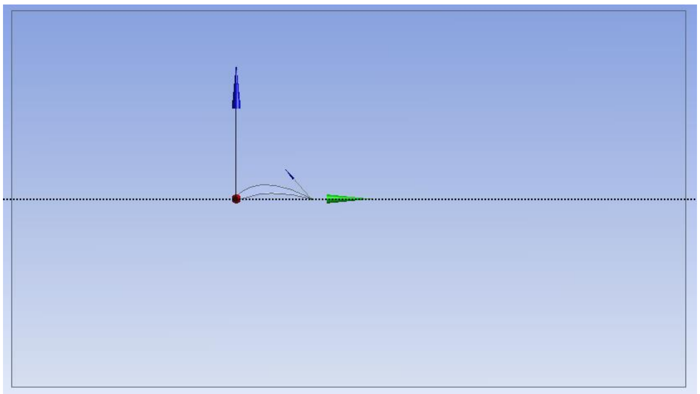

Figure 12. The optimized airfoil model in Ansys

6. Results and Discussion

In order to keep the Reynolds number as low as possible and convenient to be calculated in Ansys by using the equation $Re = \frac{\rho v L}{\mu}$, both the density and the viscosity are set to 1 as reference values in Ansys. In all experiments, the inlet velocity is set to 40, so the Reynolds number will remain unchanged at 40. Similarly, the Reynolds number is set to 40 to keep consistent.

The first case is to keep the camber=13% (camber/cord) and camber location (camber location/cord) =40% unchanged and adjust the value of thickness of the basic airfoil. The detailed results of the relationship between thickness and lift-to-drag ration are shown in Table 2 and Figure 13.

Table 2. Data of thickness and lift-to-drag ratio from Javafoil and Ansys

Cl/Cd; AOA (°) / Thickness (%)	1 (Javafoil)	2 (Javafoil)	3 (Javafoil)	4 (Javafoil)	5 (Javafoil)	6 (Javafoil)	4 (Ansys)
1	1.595	1.530	1.431	3.487	1.215	0.340	3.150
2	1.552	1.491	1.424	5.115	2.123	0.713	5.090
3	1.494	1.465	1.400	5.630	3.628	1.723	5.373
4	1.450	1.440	5.033	6.607	4.360	2.830	5.891
5	1.416	1.417	5.334	1.289	5.008	3.508	6.210
6	1.389	1.394	1.334	1.272	1.233	4.122	6.540
7	1.358	1.372	1.305	1.255	1.262	4.458	1.892
8	1.331	1.349	1.282	1.248	1.195	1.171	1.932
9	1.307	1.315	1.266	1.233	1.192	1.157	1.205
10	1.281	1.293	1.249	1.213	1.174	1.139	1.190
11	1.257	1.258	1.229	1.194	1.151	1.122	1.067
12	1.237	1.272	1.210	1.183	1.143	1.110	0.995

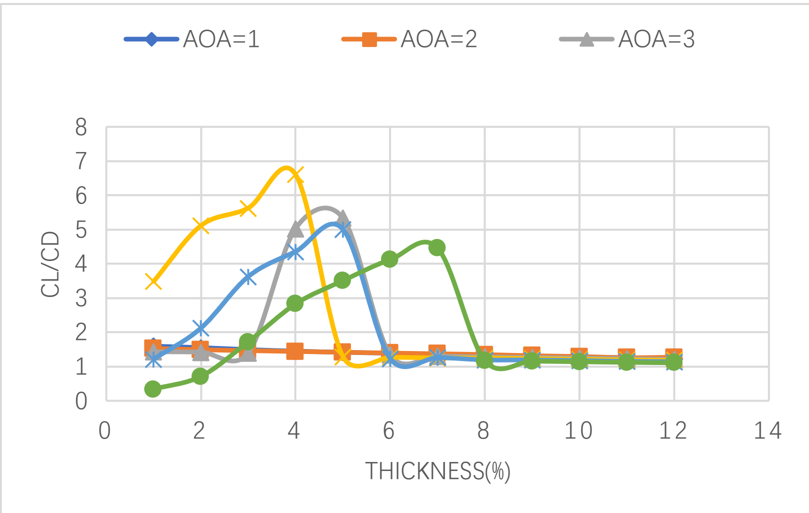

Figure 13. Diagram of thickness and lift-to-drag ratio from Javafoil

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

According to the relationship of thickness and lift-to-drag ratio shown in Javafoil, when the angle of attack is 4°, thickness of 4.9% (thickness/cord) is considered as the most optimal parameter of the airfoil. Then angle of attack=4°, camber=13%, and camber location=40% are set as reference values in Ansys, airfoils with different thickness are simulated to get the result in Figure 14.

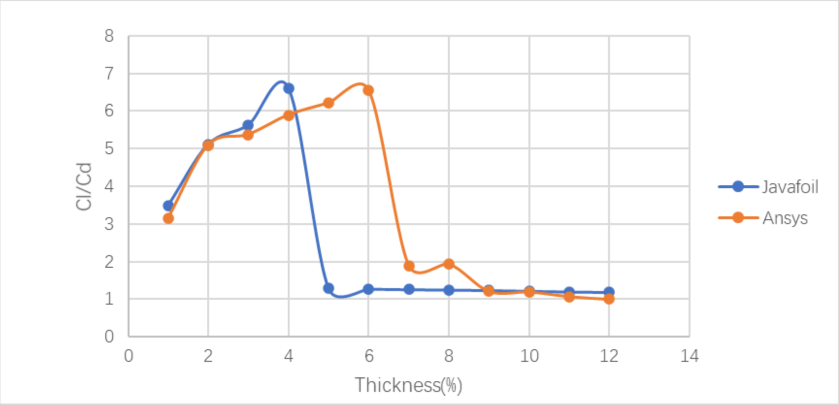

Figure 14. Diagram of thickness and lift-to-drag ratio from Javafoil and Ansys when the angle of attack=4°

According to the data in Table 2 and Figure 14, the difference error is $(5.1-4.9)/4.9\times 100\% = 4.1\%$ which can be accepted.

The second case is to keep the camber=13% (camber/cord) and thickness =4% (thickness/cord) unchanged and adjust the value of camber location of the basic airfoil. The detailed results of the relationship between camber location and lift-to-drag ratio are shown in Table 3 and Figure 15.

Table 3. Data of camber location and lift-to-drag ratio from Javafoil and Ansys

AOA (°) / Cl/Cd / Camber Location (%)	2 (Javafoil)	4 (Javafoil)	6 (Javafoil)	7 (Javafoil)	8 (Javafoil)	9 (Javafoil)	10 (Javafoil)	9 (Ansys)
1	4.586	4.950	10.902	0.118	0.102	0.086	0.078	0.075
2	5.605	8.458	8.020	0.324	1.771	1.479	1.256	1.398
3	2.361	4.483	5.767	3.334	0.412	3.649	3.076	3.776
4	1.926	4.649	11.334	13.19	14.223	23.589	10.73	4.680
5	1.592	4.285	9.804	12.536	14.253	14.534	14.413	24.003
6	1.444	4.484	10.332	13.409	15.078	15.623	0.405	13.886
7	1.322	4.299	9.996	0.494	14.061	14.683	0.380	12.697
8	1.280	4.363	10.174	12.303	12.474	12.454	0.423	12.310
9	1.168	4.535	10.639	11.739	0.693	0.548	0.497	0.637
10	0.590	4.950	10.902	11.264	0.574	0.542	11.046	0.536

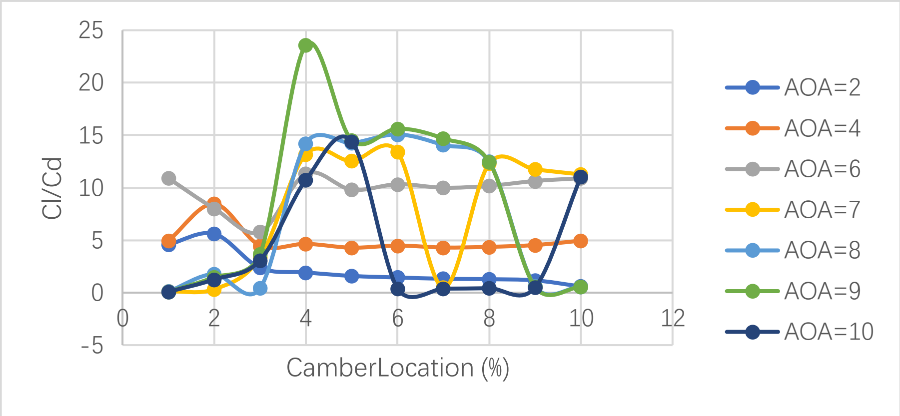

Figure 15. Diagram of camber location and lift-to-drag ratio in Javafoil

International Core Journal of Engineering Volume 7 Issue 1, 2021
ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

According to the relationship of camber location and lift-to-drag ratio, when the angle of attack is 9°, camber location of 4.8% (camber location/cord) is considered as the most optimal parameter of the airfoil. Then the angle of attack=9°, camber=13%, and thickness=4% are set as reference values in Ansys, airfoils with different camber location are simulated to get the result in Figure 16.

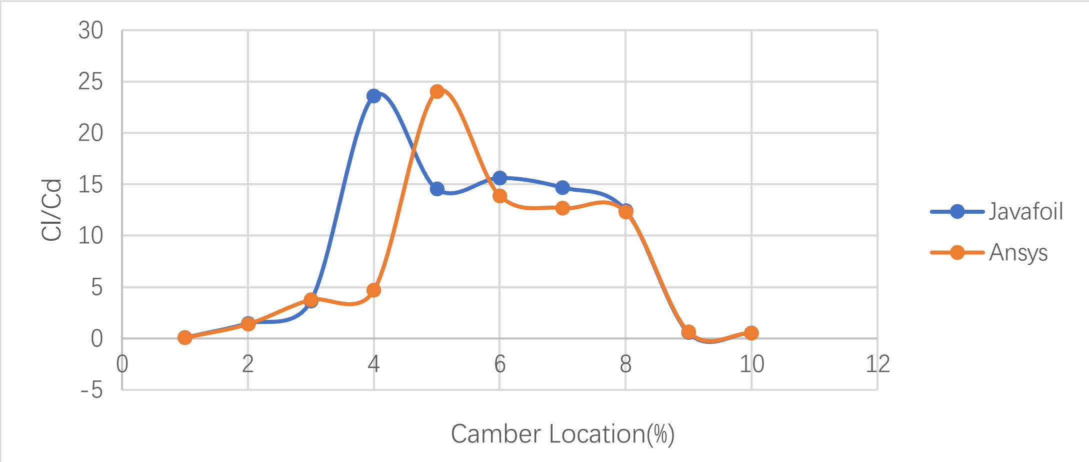

Figure 16. Diagram of camber location and lift-to-drag ratio from Ansys and Javafoil when the angle of attack =9°

According to the data in Table 3 and Figure 16, the difference error is (5.0-4.8)/4.8*100%=4.2% which can be accepted.

The third case is to keep the camber location=40% (camber location/cord) and thickness =12% (thickness/cord) unchanged and adjust the value of camber of the basic airfoil. The detailed results of the relationship between camber and lift-to-drag ration are shown in Table 4 and Figure17.

Table 4. Data of camber and lit-to-drag ratio from Javafoil and Ansys

Cl/Cd AOA (°) Camber (%)	0 (Javafoil)	2 (Javafoil)	4 (Javafoil)	6 (Javafoil)	0 (Ansys)
0	0	0.3	0.544	0.743	0
1	0.183	0.452	0.670	0.892	0.170
2	0.361	0.591	0.782	0.931	0.325
3	0.528	0.717	0.899	0.964	0.480
4	0.682	0.846	0.959	1.021	0.579
5	0.908	0.917	1.029	1.067	0.999
6	1.043	0.997	1.084	1.102	1.168
7	1.147	1.060	1.124	1.129	1.319
8	1.225	1.187	1.147	1.145	1.328
9	1.282	1.230	1.164	1.146	1.334
10	1.332	1.246	1.169	1.147	1.346
11	1.358	1.253	1.189	1.137	1.357
12	1.371	1.240	1.193	1.125	1.389
13	1.375	1.256	1.183	1.110	1.375
14	1.367	1.222	1.158	1.090	1.362
15	1.359	1.203	1.139	1.070	1.350
16	1.349	1.185	1.144	1.048	1.300
17	1.330	1.160	1.117	1.029	1.250
18	1.306	1.134	1.095	1.043	1.215
19	1.276	1.112	1.071	1.011	1.190
20	1.258	1.089	1.045	0.993	1.130

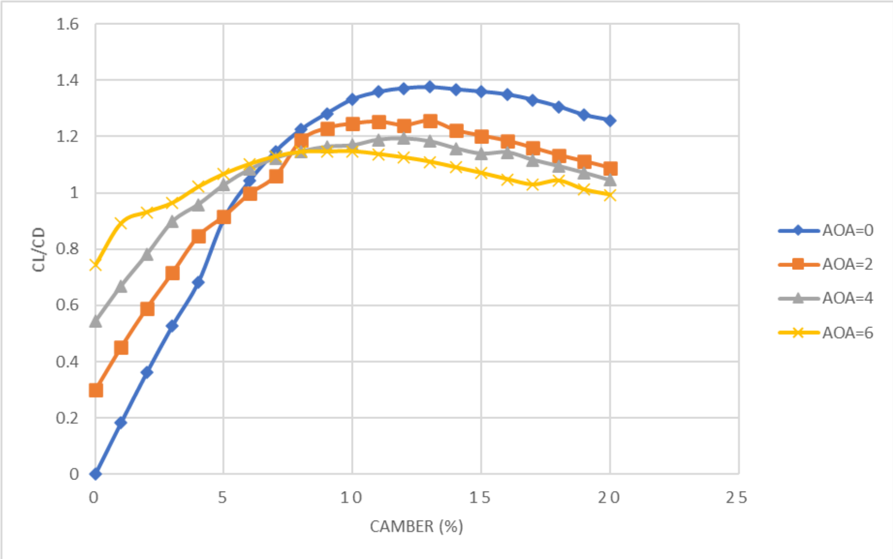

Figure 17. Diagram of camber and lift-to-drag ratio from Javafoil

According to the relationship of camber and lift-to-drag ratio, when the angle of attack is 0°, camber location of 12.1% (camber /cord) is considered as the most optimal parameter of the airfoil. Then the angle of attack=0°, camber location=40%, and thickness=12% are set as reference values in Ansys, airfoils with different camber are simulated to get the result in Figure 18.

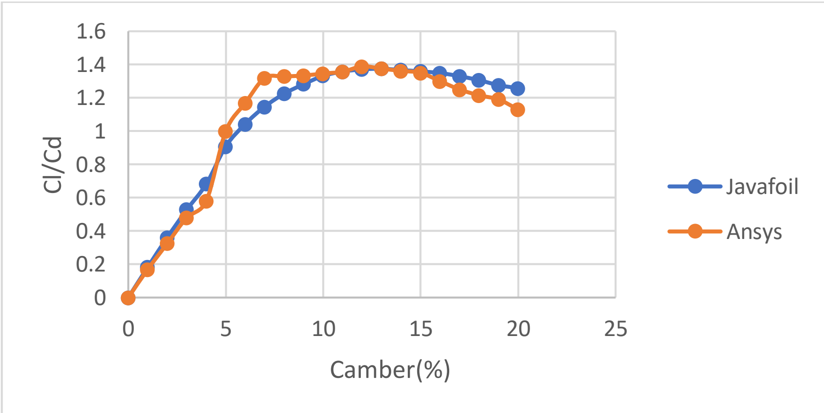

Figure 18. Diagram of camber and lift-to-drag ratio from Javafoil and Ansys when the angle of attack=0°

According to the data in Table 4 and Figure 18, the difference error is (12.1-12)/12.1*100%=0.8% which can be accepted.

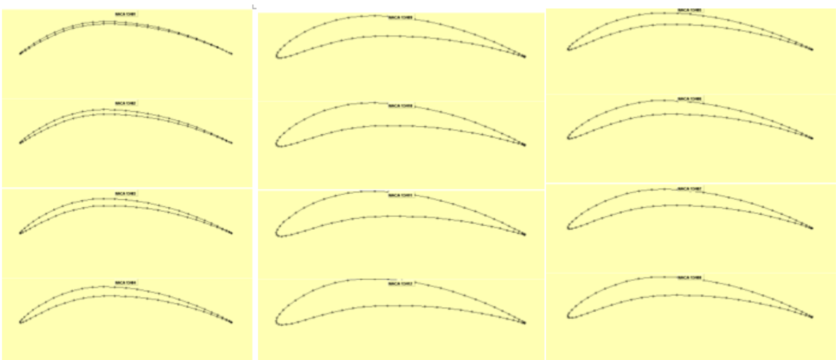

Figure 19. Different airfoil shapes when thickness is different

ISSN: 2414-1895
DOI: 10.6919/ICJE.202101_7(1).0046

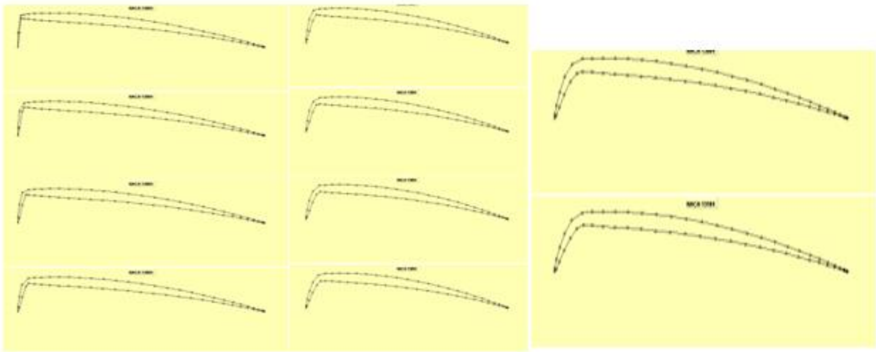

Figure 20. Different airfoil shapes when the camber location is different

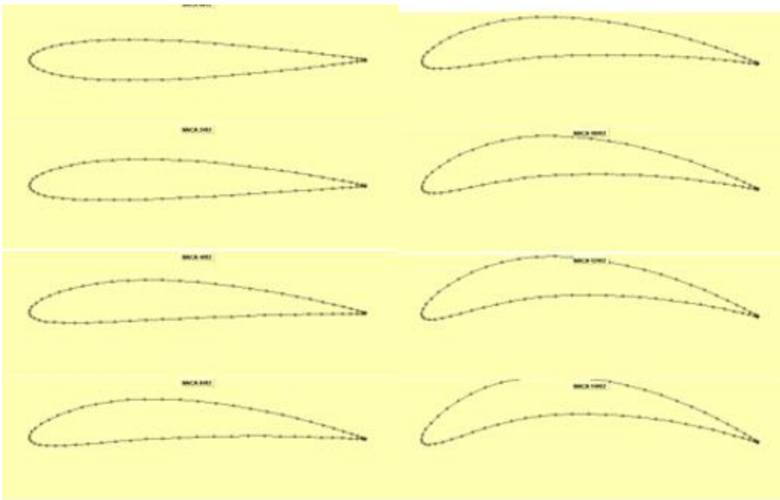

Figure 21. Different airfoil shapes when the camber is different

7. Conclusion

Lift-to-drag ratio represents for the aerodynamic efficiency of a type of airfoil. By utilizing the software of Javafoil and Ansys Fluent, we create models of airfoils that vary in camber, camber location or thickness at different angle of attack and do simulations to output the lift and drag coefficients. This research clearly brings out the importance to create a database for high performance airfoils for low Re applications [7]. In this work, an optimization study has been carried out for the three parameters of the airfoil shape, and different shapes have been obtained. It is worthwhile to extend the study to find an optimal shape that gives good performance for higher Reynolds number. This work is also being extended to find optimal geometries for unsteady flows [7]. Our work set a reference for the values of lift-to-drag ratio of various airfoils at different angle of attack and is considered valuable for further study on the complex wing design which aims at improving the performance of types of aircrafts.

References

[1] Khalid, A., & Kumar, P. (2014). Aerodynamic Optimization of Box Wing – A Case Study. International Journal of Aviation, Aeronautics, and Aerospace, 1(4). Retrieved from https://commons.erau.edu/ ijaaa/ vol1/iss4/6 [Accessed 18 July 2020]

[2] Zhang, T., Huang, W., Wang, Z. et al. A study of airfoil parameterization, modeling, and optimization based on the computational fluid dynamics method. J. Zhejiang Univ. Sci. A 17, 632–645 (2016).

[3] Gudmundsson, S. (2014). General Aviation Aircraft Design: Applied Methods and Procedures: Vol. First edition. Butterworth-Heinemann.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202101_7(1).0046

[4] Olivier Cleynen. (2011). Vocabulary terms used in the aeronautical domain. Available at: https://zh. wikipedia.org/wiki/File:Wing_profile_nomenclature.svg [Accessed 10 July 2020]

[5] Deng, S., Jiang, C., Wang, Y., & Wang, H. (2018). Helicopter Flight Dynamics Simulation with Continues -Time Unsteady Vortex Lattice-Free Wake and Multibody Dynamics. The Proceedings of the 2018 Asia-Pacific International Symposium on Aerospace Technology (APISAT 2018).

[6] Jeong-Hoo, Park, Se-Myong, Chang, Shin-Young, & Lee. (2014). Drag coefficient and friction factor on the fundamental components for given surface roughness.

[7] Srinath, D. N., & Mittal, S. (2010). Optimal airfoil shapes for low Reynolds number flows. International Journal for Numerical Methods in Fluids, 61(4), 355-381.

[8] Viola, I. M., Chapin, V., Speranza, N., & Biancolini, M. E. (2018). Optimal airfoil's shapes by high fidelity CFD. Aircraft Engineering and Aerospace Technology, 90(6), 1000-1011.