International Core Journal of Engineering Volume 6 Issue 8, 2020
ISSN: 2414-1895 DOI: 10.6919/ICJE.202008_6(8).0033

2D Optimization of NACA Airfoil with PSO

--Under ANSYS Circumstance

Yilong Li[1,a], Shiyi Lu[2,b], Hengyuan Yang[3,c]

[1]Harbin Institute of Technology, Harbin 150001, China;

[2]Beihang University, Haidian, Beijing 100191, China;

[3]Rensselaer Polytechnic Institute, Troy, NY, 12180, US.

[a]118132505006@outlook.com, [b]lushiyi@buaa.edu.cn, [c]hengyuanyang0105@gmail.com

Abstract

The design of an airfoil is mainly analysed in two ways, structural and aerodynamic approaches. This paper focuses on the aerodynamic analysis of airfoil. The purpose is to obtain an optimized 2D airfoil shape with ideal ratio of drag and lift force. The modelling and simulation are realized using MATLAB programming and Ansys-Fluent program. The results are then analysed by PSO (Particle Swarm Optimization) in order to find the ideal shape parameters for the airfoil which has the optimal ratio of drag and lift force.

Keywords

PSO, NACA airfoil, ANSYS, 2 dimensional optimization, MATLAB.

1. Introduction

The wings of an aircraft are the most essential component that lift it up, and therefore the evaluation of the ratio between air drag force and lift force is indispensable in designing an aircraft. However, it is not an easy task to optimize the coefficients of drag and lift force, as many constraints, such as shape, angle of attack, need to be taken into consideration when doing the computation. In this project, the design of an airfoil is analysed in 2D environment based on cross-section shape in the XY-plane, and several dominating parameters are chosen in determining the shape of the airfoil cross-section.

In order to compute the drag and lift force, computational fluid dynamics (CFD) is indispensable in doing the simulation. Cross-section shape is determined by the following parameters chosen: camber length, maximum chord thickness, and maximum camber thickness. Angle of attack is also another constraint that being manipulated after the optimization in order to test its stall condition. Before doing the simulation for airfoils, convergence test had been done to determine the domain of fluid and the mesh size for the airfoil. To ensure the stimulation method can get accurate results, cylinder model had been used to validate this CFD method. After simulations for the airfoil, the data obtained was dealt with PSO (Particle Swarm Optimization), a computational method that optimizes a problem by iteratively trying to improve a candidate solution with regard to a given measure of quality, which will be discussed later.

2. Problem Statement

In the description of fluid motion, one of the most important equation is the Naiver-Stokes equation arisen from applying Newton’s second law. Theoretically by solving these equations, the velocity field can be figured out for viscous fluid. However, in most cases, the Naiver-Stokes equations are nonlinear partial differential equations, which means only the simplest questions can be answered by

ISSN: 2414-1895 DOI: 10.6919/ICJE.202008_6(8).0033

solving the equations using calculus methods. In this program, the velocity field of the air flowing through an airfoil is too complicated to be solved by calculus methods.

To get the lift force and drag force of an airfoil, computers must be used to solve the equations. As the theoretical equations between the parameters of airfoil and the forces applied on it are remained unknown, the common methods used to get the optimal solution such as derivation method do not work in the optimization of airfoil. Special methods or theories must be applied in this project.

3. Processing Method

This project mainly consists of three parts: modelling, simulation, and optimization. The optimized coefficients of drag and lift force, and shape parameters of airfoil were found by looping the three parts.

Modelling was done by the combined use of MATLAB and Ansys Fluent program. The shape of chord was governed by the equation below, where m represents the maximum camber, t represents the maximum thickness as a fraction of the chord and p represents the location of maximum camber. The equation was formulated in MATLAB, with airfoil parameters as variable. [1]

$$y_c = \begin{cases} \frac{m}{p^2}\left(2p\left(\frac{x}{c}\right) - \left(\frac{x}{c}\right)^2\right) & 0 \le x \le pc \\ \frac{m}{(1-p)^2}\left((1-2p) + 2p\left(\frac{x}{c}\right) - \left(\frac{x}{c}\right)^2\right) & pc \le x \le c \end{cases}$$

For the airfoil represented by this equation, in order to apply the thickness perpendicular to the camber line, the coordinates (Xu, Yu) and (Xl, Yl), of the upper and lower airfoil surface, become as following graph.

$$x_U = x - y_t \sin \sin \theta\ , \qquad y_U = y_c + y_t \cos \cos \theta$$
$$x_L = x + y_t \sin \sin \theta\ , \qquad y_L = y_c - y_t \cos \cos \theta$$

where,

$$\theta = \left(\frac{d_{yc}}{dx}\right)$$

$$\frac{dy_c}{dx} = \begin{cases} \frac{2m}{p^2}\left(p - \frac{x}{c}\right), & 0 \le x \le pc \\ \frac{2m}{(1-p)^2}\left(p - \frac{x}{c}\right), & pc \le x \le c \end{cases}$$

The model used in stimulation part is the SST k-ω turbulence model which is a two-equation eddy-viscosity model.[2] The use of a k-ω formulation in the inner parts of the boundary layer makes the model directly usable all the way down to the wall through the viscous sub-layer.[3] It is a hybrid model combining the Wilcox k-omega and the k-epsilon models. A blending function, F1, activates the Wilcox model near the wall and the k-epsilon model in the free stream. This ensures that the appropriate model is utilized throughout the flow field.[2]

In the optimization part, PSO (Particle Swarm Optimization) is utilized to obtain a set of optimized parameters for the airfoil chosen. PSO is a computational method initialized from imitating a bird flock or fish school. Assuming birds searching food in a certain area, but they do not know where the food is. With the distance between the birds and the food known, the optimal strategy to find the food

ISSN: 2414-1895
DOI: 10.6919/ICJE.202008_6(8).0033

is to search the surrounding of the bird nearest to the food. The basic loop of optimization is as described in the following figure. [4]

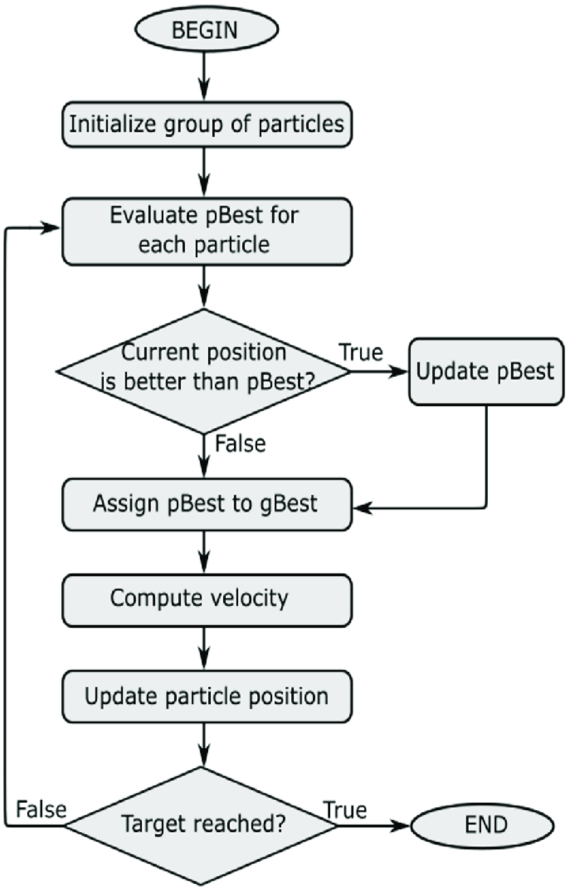

Fig. 1: Procedure of PSO algorithm

In this project, chord length is defined as a constant and ‘t’, ‘p’, ‘m’ in the equation above as three controlled parameters used to generate airfoils. Therefore, the best combination of (t, p, m) can be regarded as a point in the three-dimensional space and PSO algorithm can be used to find this point.

4. Validation

4.1 Parameters and Constants Defined

Reynold number: $Re = \frac{\rho VD}{\mu}$, V is the inflow velocity, μ is the viscosity of air

Average drag coefficient: $C_d = \frac{F_d}{0.5\rho V^2 HD}$, Fd is the drag force applied on the cylinder, H is transverse length of cylinder (1m in the tests)

The density of air: ρ=1.225kg/m^3

The viscosity of air: μ=1.8375*10^{-5}kg/m-s

4.2 The Result of Validation

Considering when Reynold Number is less than 40, the fluid flow can be seen as a laminar flow, the result from a book [1] about the analysis of cylinder streaming was used as the standard Cd. An empirical formula was used for high Reynold number situation (the research from Brown and Lawler) [5]-[7].

The result is showed in the table below

$$C_d = \frac{24}{Re}(1 + 0.15Re^{0.681}) + \frac{0.407}{1 + 8710Re^{-1}}$$

ISSN: 2414-1895
DOI: 10.6919/ICJE.202008_6(8).0033

Table 1: Result of validation of different Re

Reynold Number	Velocity/m*s-2	Cd	Standard Cd	Source of Standard Cd
20	0.012	2.389	2.400	essay
100	0.06	1.329	1.470	empirical formula
3900	2.34	0.536	0.671	empirical formula
1000000	600	0.447	0.466	empirical formula
10000000	6000	0.443	0.431	empirical formula

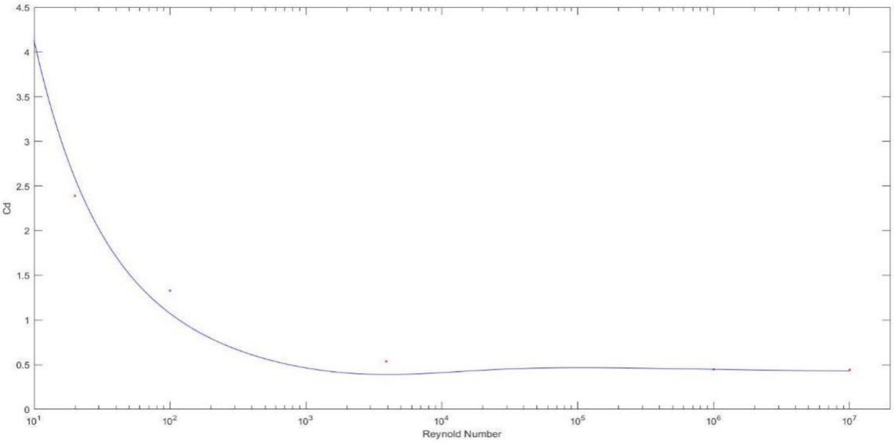

Fig. 2: Cd values by empirical formula and by simulation

As being showed in the table and figure above, the curve in figure 1 represents the Cd by empirical formula, and the dots represent the Cd values gained by simulation. When the Reynold Number was large enough (the condition airfoil works in), the method chosen perfectly satisfied the empirical formula. This situation confirms that this model can be used for the project.

The figures below show the flow field under different Reynold Numbers. Figure 2's Reynold Number is 20 and it can be seen as the laminar flow. Figure 3's Reynold Number is $1*10^7$ and it stimulates the real condition the airfoil works in.

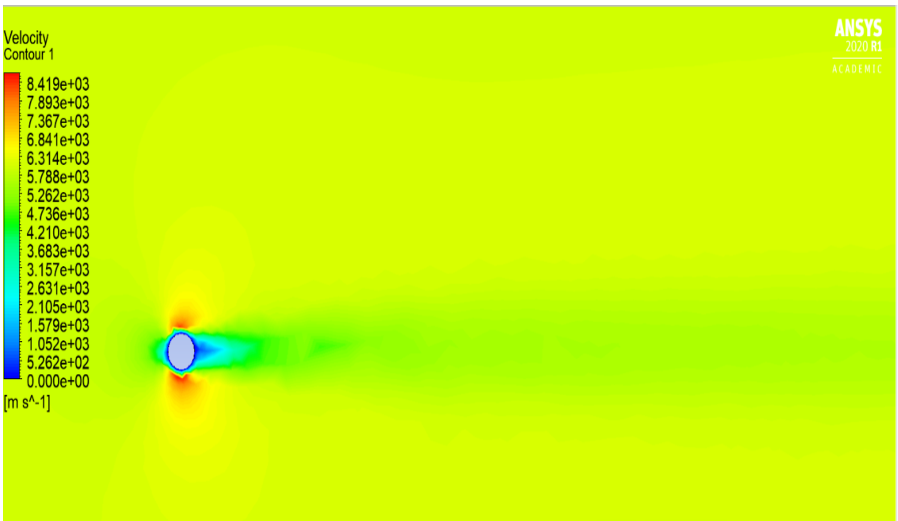

Fig. 3: Cylinder simulation with Re = 20

ISSN: 2414-1895
DOI: 10.6919/ICJE.202008_6(8).0033

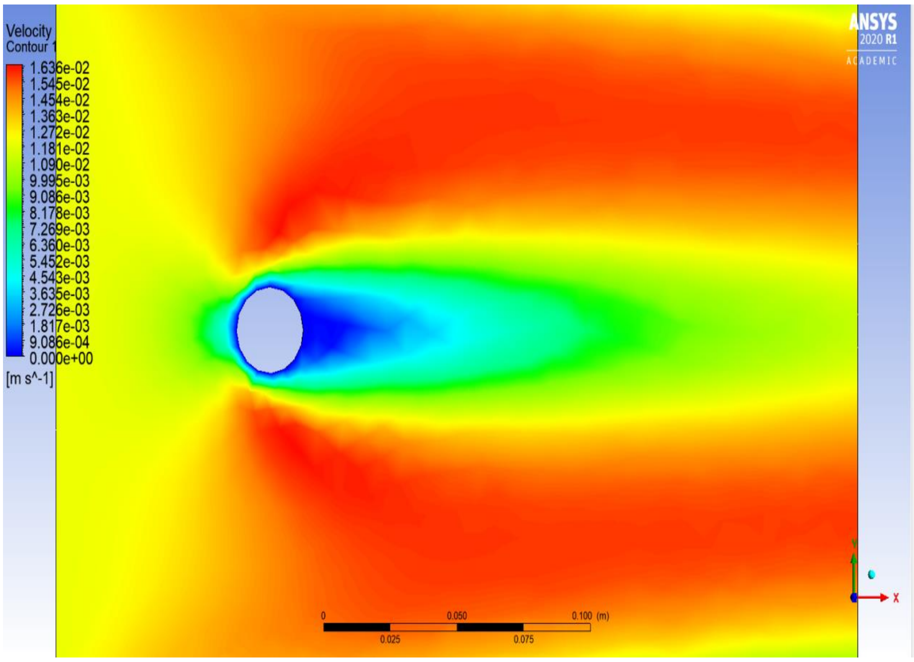

Fig. 4: Cylinder simulation with Re = $1*10^7$

5. Convergence Test

In the fluent simulation, besides the theoretical model discussed in the validation part, the sizes of both domain and mesh also make significant impact on the result exported by Ansys. The affection brought by them are showed in the figures below. The first one shows the case when the fluid domain was not large enough for the simulation. Because the upper and lower boundaries were set as walls in the computation, the boundaries limited the flow of the air and caused the theoretical error in the program. The second one shows the case when the mesh size was not as tiny as we can ignore its affection, which caused the smooth curve of the airfoil to be cut into series of broken lines. Both situations are not supposed to happen in this program.

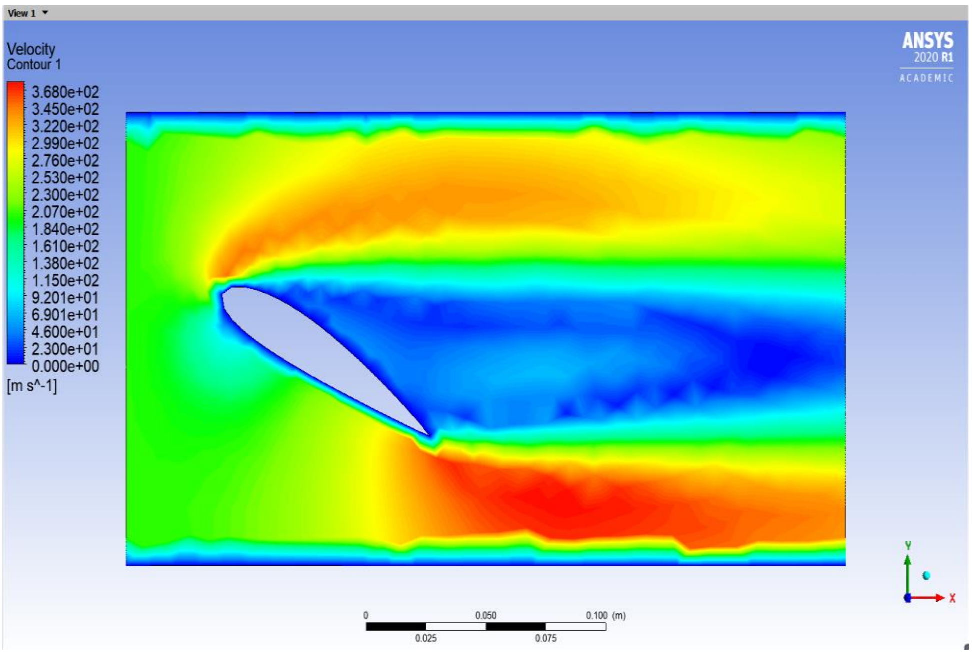

Fig. 5: Simulation done when the fluid domain is not large enough

ISSN: 2414-1895 DOI: 10.6919/ICJE.202008_6(8).0033

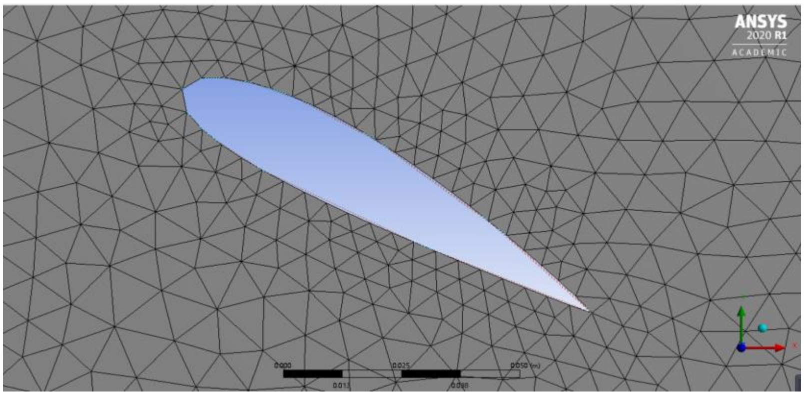

Fig. 6: Mesh size not tiny enough to ignore its effect

5.1 The Convergence of Domain Size

To make sure the domain is large enough, the boundaries were set to begin with a small scale and then expand until the result of simulation became stable. In the whole program, the length of chord was fixed to 100mm and the relative velocity between air and airfoil was set at 200m/s. Considering the increase in attack angle, the fluid required larger domain to become steady, the greatest attack angle was decided to be 30 degrees in the convergence test. The coefficients of drag and lift were used as the criteria of the convergence test.

After 8 tests, when the height of domain reached 500mm (200mm above the chord and 300mm below the chord), the results were 2754.3 (drag coefficient) and 5301.9 (lift coefficient) respectively. Then the domain height was expended to 1100mm (500mm above the chord and 600mm below the chord). The results changed to 2696.0 (drag coefficient) and 5310.8 (lift coefficient). The difference between the two sets of results is within 5%. It can be easily concluded that the simulation can be completed properly when the height of the domain is larger than 500mm (5 times as the length of chord).

Using the same method for the domain height test, when the length of domain was doubled from 300mm to 600mm, the result did not change, turning out a large scale of domain length is not required for the analysis.

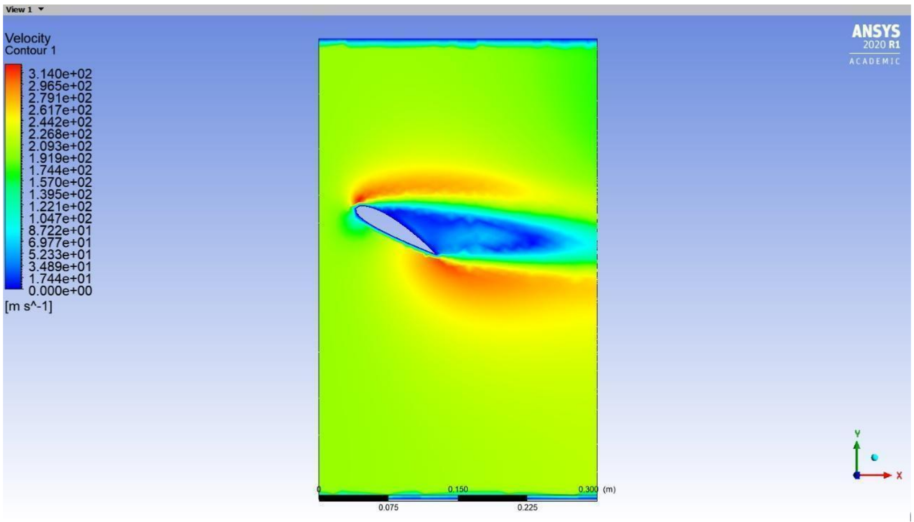

Fig. 7: Simulation done when the fluid domain is large enough

5.2 The Convergence of Mesh Size

The convergence of mesh size began with a relatively large value: 0.01mm for the main body and 0.005mm for the airfoil edge. When the mesh size decreased to 0.005mm for the main body and

ISSN: 2414-1895
DOI: 10.6919/ICJE.202008_6(8).0033

0.0025mm for the airfoil edge, the resulted number started converging. In this situation, the obtained coefficients were 2555.0 for drag and 5390.3 for lift. As the size reduced by half (0.0025 for main body and 0.0012 for the airfoil edge), the number changed to 2430.6 for drag and 5490.5 for lift respectively. The difference between the two sets of results were within 5%.

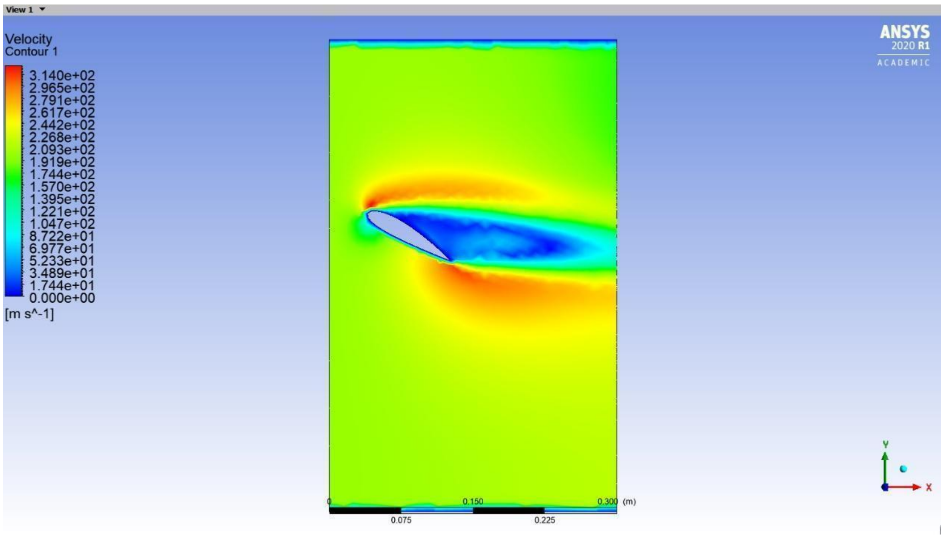

Fig. 8: Final domain size

5.3 The Geometry Size of the Optimization

As a result, the domain of fluid is determined to be 500mm*300mm, and the mesh sizes are 0.005 for the main body and 0.0025 for the airfoil edge.

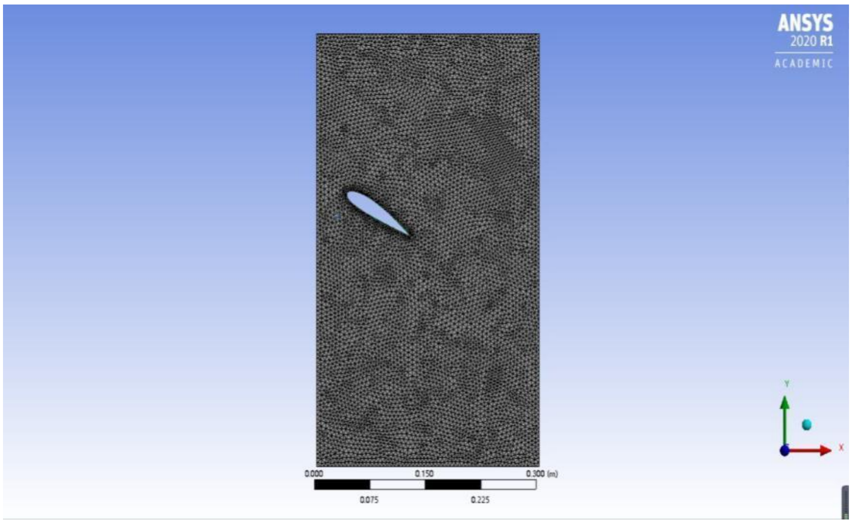

Fig. 9: Final mesh size

6. Simulation Results and Data Analysis

6.1 The Optimization of Airfoil Parameters

The parameters optimization is based on zero angle of attack condition. In this part, nine airfoils were defined as a group and six generations were calculated. During each generation, the parameters were updated automatically by PSO algorithm. Every time the parameters change, simulation will be conducted and the optimal solution (the airfoil with maximum L/D) in this group will be found.

ISSN: 2414-1895
DOI: 10.6919/ICJE.202008_6(8).0033

The initial points of the program were chosen form 9 existing NACA airfoils.[1] By importing the models into Ansys, the simulation results of lift force and drag force are obtained. Inputting the parameters of the nine airfoils and the L/D into the program, nine groups of new airfoil parameters will be given by the program. Using the new parameters to generate airfoils and get the L/D through Ansys simulation, the maximum L/D in every generation indicates whether the optimization is going well. Keep running the program and the simulation result of optimal solution in every generation is showed below.

Table 2: The parameters of optimal solution in each generation

Generation number	1	2	3	4	5	6
t	0.12	0.073	0.089	0.077	0.077	0.07575
p	0.4	0.36	0.268	0.407	0.411	0.405
m	0.05	0.049	0.016	0.0526	0.0527	0.0519
L/D	20.0	20.24	20.42	21.07	20.92	21.37

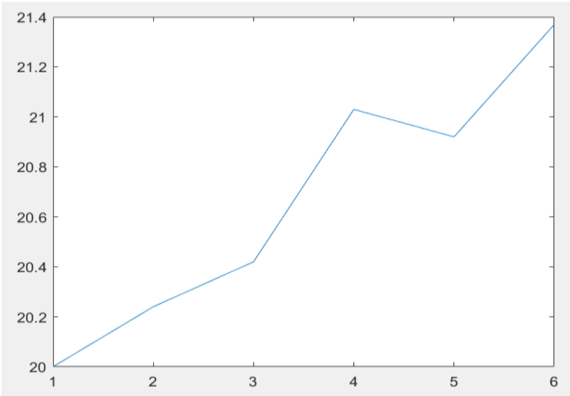

Fig. 10: The variation tendency of L/D

The velocity field of optimal solution in each generation:

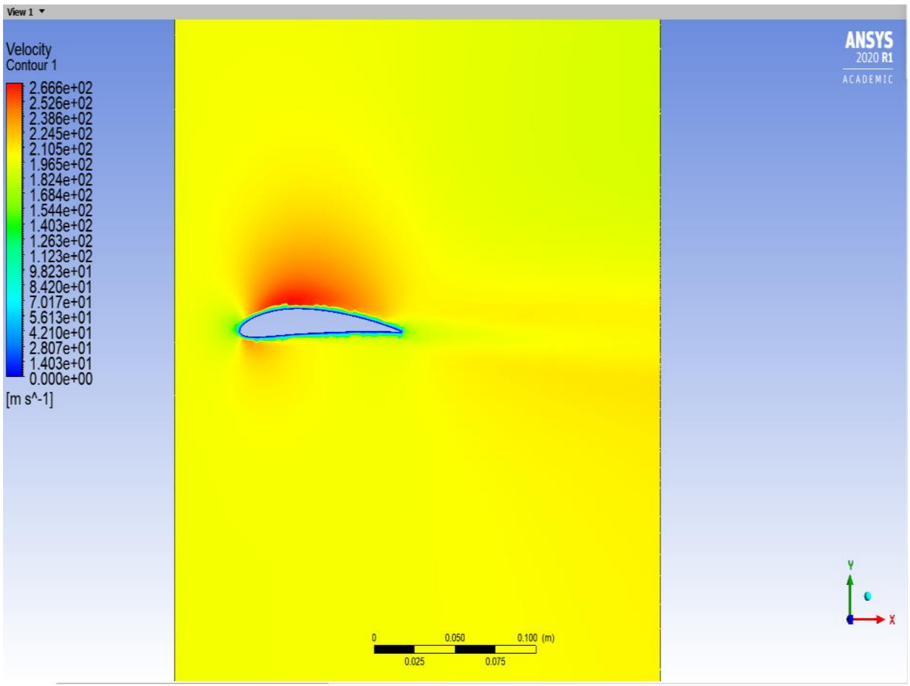

Fig. 11: Generation 1

ISSN: 2414-1895
DOI: 10.6919/ICJE.202008_6(8).0033

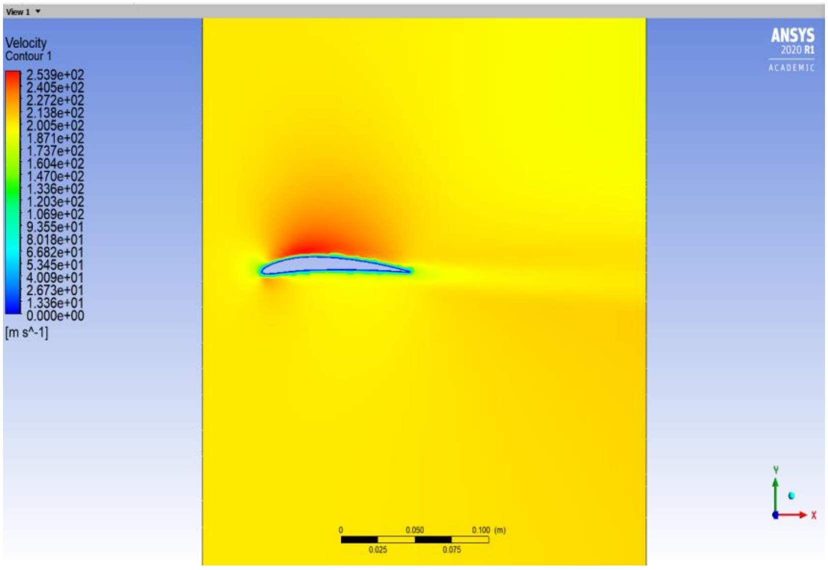

Fig. 12: Generation 2

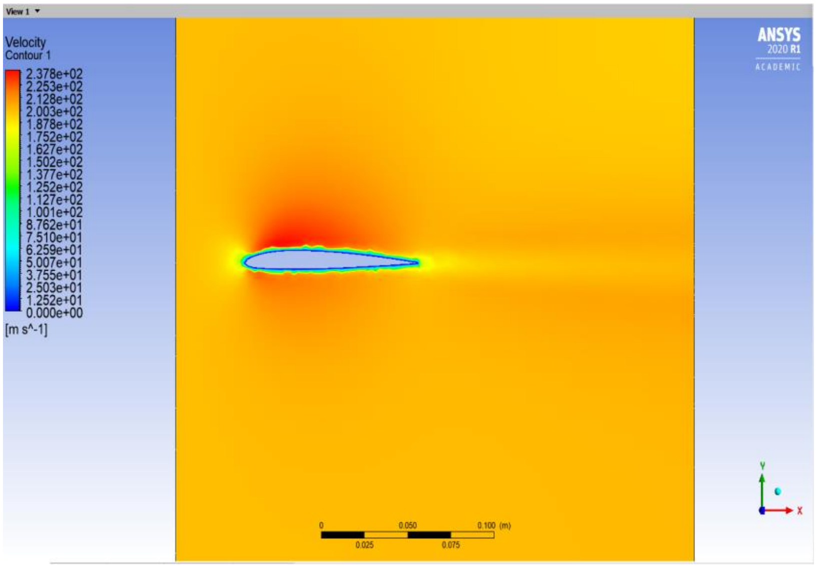

Fig. 13: Generation 3

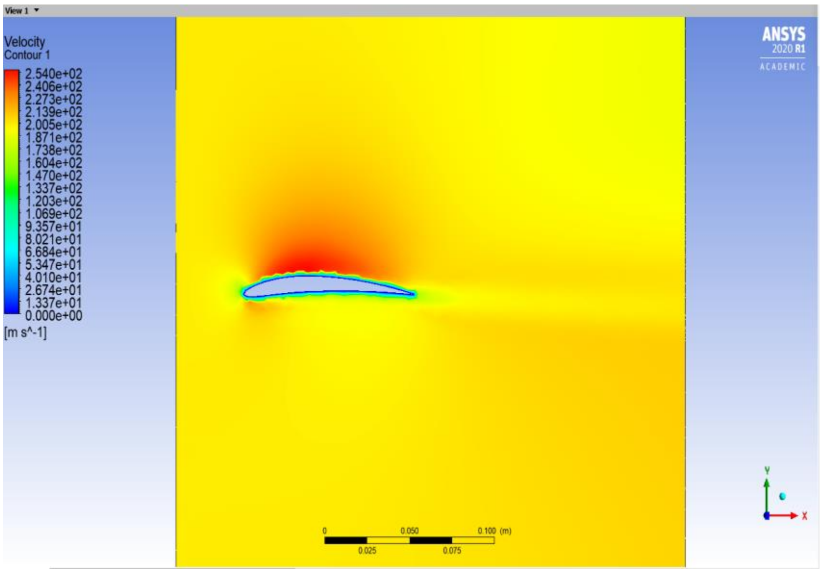

Fig. 14: Generation 4

ISSN: 2414-1895
DOI: 10.6919/ICJE.202008_6(8).0033

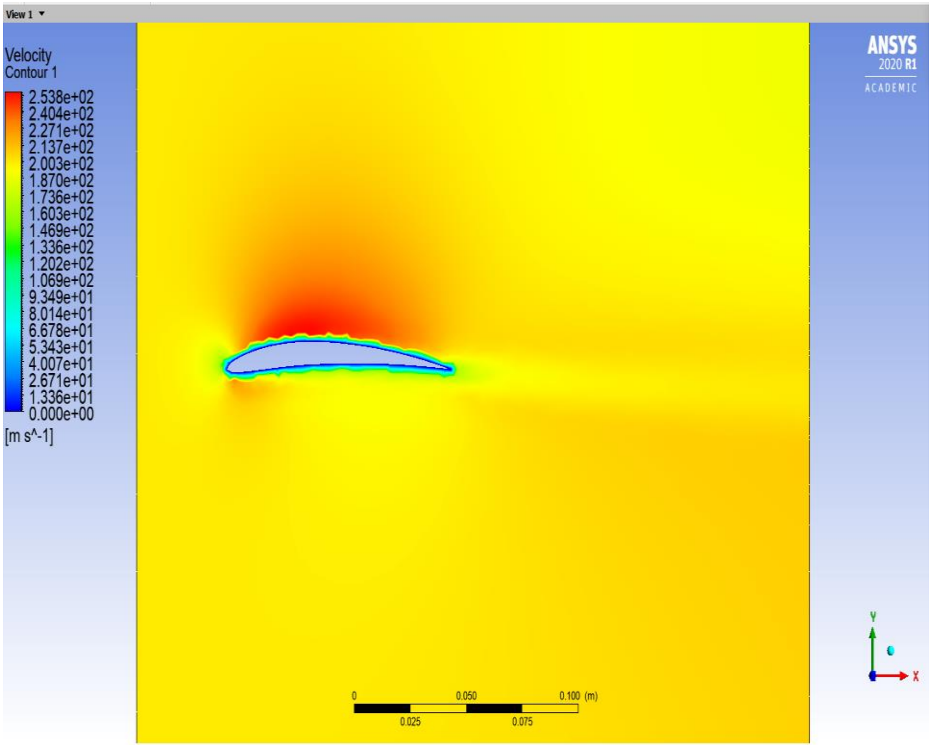

Fig. 15: Generation 5

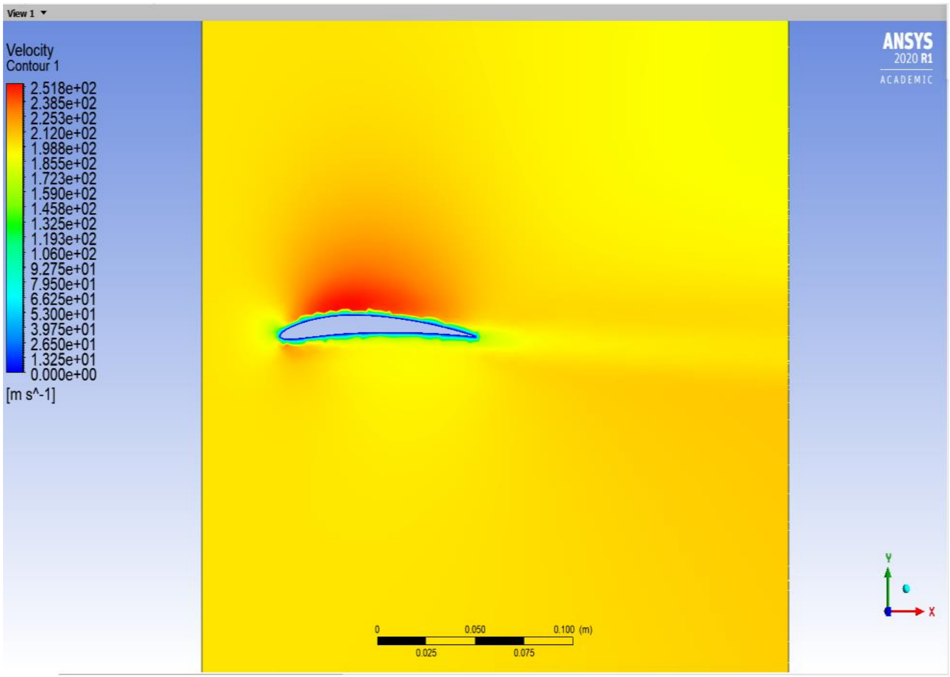

Fig. 16: Generation 6

It is obvious that the L/D grew steadily from 20.0 to 21.37 after 6 generations. Although there was a slight decrease in generation 5, the ratio rose dramatically to 21.37 in generation 6. Generally speaking, the algorithm is functioning.

6.2 The Analysis of Stall Phenomenon

In fluid dynamics, a stall is a reduction in the lift coefficient generated by a foil as angle of attack increases. This occurs when the critical angle of attack of the airfoil is exceeded. [8] Stalls in fixed-wing flight are often experienced as a sudden reduction in lift as the pilot increases the wing's angle

ISSN: 2414-1895 DOI: 10.6919/ICJE.202008_6(8).0033

of attack and exceeds its critical angle of attack (which may be due to slowing down below stall speed in level flight).[9]

To ensure the security of the working airfoil, the stall attack angle at 200m/s was calculated in the program. By using ANSYS to simulate the working situation under different attack angle, the lift force applied on the airfoil was calculated and showed in the table and graph below.

Table 3: Relation between attack angle and lift force

Attack angle/°	**0**	**5**	**10**	**11.5**	**12**	**12.5**	**13**	**15**	**20**
Lift force/N	1233.6	2267.5	3208.8	3380	3480.9	3409.8	3334.8	3265.7	3126.5

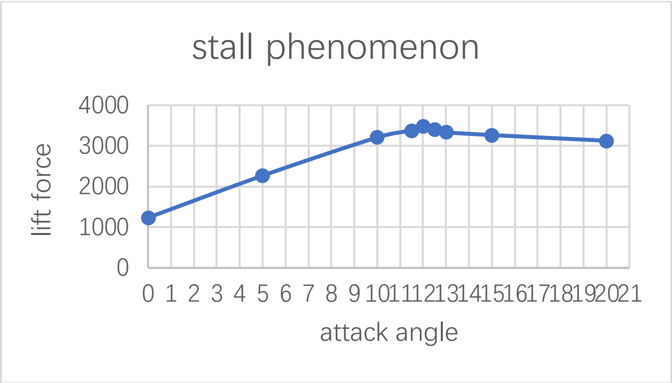

Fig. 17: Lift force applied on the airfoil for different attack angles

The graph shows that the greatest amount of lift is produced as the attack angle is around 12 degrees. When the attack angle is larger than 12 degrees, the airfoil produces less lift force as the angle continuously increases. Thus the stall angle for this airfoil is 12 degrees.

The stall is caused by flow separation which, in turn, is caused by the air flowing against a rising pressure. The figure below shows the streamline of air when the attack angle is 20 degrees. The separated flow causes buffeting and is so dominant that the lift force falling from its peak value to 3126.5N (89.8% of the peak value).

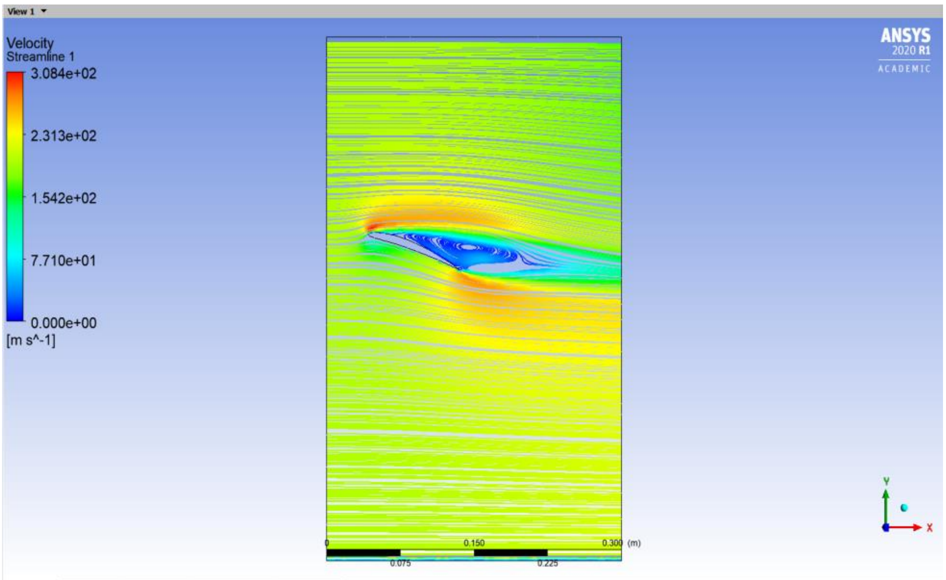

Fig. 18: The streamline of air when the attack angle is 20 degrees

ISSN: 2414-1895 DOI: 10.6919/ICJE.202008_6(8).0033

7. Conclusion

The article proposed an approach based on PSO algorithm to conduct 2D airfoil optimization for NACA airfoil system. The destination of optimization is to get the airfoil shape which has the greatest ratio of lift and drag. After six generations of iterative computation, the L/D increased from 20 to 21.37, which indicates that this approach is practical.

Although the optimization is going well, there are still a few problems in the whole process. During the optimization of airfoil parameters, the maximum L/D was expected to increase every generation. The results from generation 1 to generation 4 were ideal. But the L/D decreased in generation 5. The reason for this phenomenon seems to be multiple. The most obvious one should be the lack of sample. In PSO algorithm, the number of particles in a group is usually defined as 50 or 100, but there are only nine airfoils in a group in this program. Therefore, it is possible that all the sample points are moving towards the wrong direction in a generation. Local optimal solution may also lead to this phenomenon. The PSO algorithm has the ability to escape from local optimal solution, but it may also be trapped by local optimal solution for a few generations. Focusing on the graph of L/D (fig 11), it still has a tendency to increase.

For future study in this field, the attention can be paid to the automation of simulation. Instead of doing most of the work artificially, using programming method to let ANSYS read the 2D curves of airfoil from MATLAB automatically and do the stimulation work following the process defined would remarkably decrease the workload of researchers and make it possible to increase the quantity of individuals in each generation, which may probably provide a better result in optimization. Also, with automated work, this method of optimization of airfoil can be directly used in other problem in fluid dynamic field.

References

[1] NACA airfoil - Wikipedia. Retrieved from: https://en.wikipedia.org/wiki/NACA_airfoil

[2] Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", 1994, AIAA Journal, vol. 32, no 8. pp. 1598-1605.

[3] [3] Hermann Schlichting. Boundary layer Theory [M]. New York: McGraw-Hill Book Company, 1979.

[4] Shi Y H, Eberhart R C . Empirical study of particle swarm optimization[C]// Congress on Evolutionary Computation-cec. IEEE, 2002.

[5] BROWNPP, LAWLERDF. Sphere drag and setting velocity revisited[J], Journal of Environmental Engineering - ASCE, 2003(3) : 222 – 231.

[6] CLIF R, GRACE JR, WEBER M E. Bubbles, drops, and particles[M]. Academic Press，New York, 1978.

[7] TURTON R, LEVENSPIEL O. A short note on the drag correlation for spheres[J]. Powder Technology, 1986, 47 (1): 83 - 86.

[8] USAF & NATO Report RTO-TR-015 AC/323/(HFM-015)/TP-1, 2001

[9] Anderson, J.D., A History of Aerodynamics. Cambridge University Press. ISBN 0-521-66955-3, 1997.