

# MIT - 2.094 - Term Project - Hydrofoil analysis using CFD

*Ofer Aharon*  
*Naval Construction and Engineering Program*  
*Department of Mechanical Engineering*  
*MIT, Cambridge, MA 02139, USA*

*5/1/2008*

## **Abstract**

This project describes the influence of two sequential foils on each other when are very close horizontally. The solution was achieved using ADINA. In order to verify the process, a comparison was made between analytical, experimental and ADINA's results for one foil with several angles of attack. The matching and accuracy obtained are very high. During the project, the offsets of the foil were corrected due to non-physical offsets given at the leading edge. In addition, some literature was reviewed (see references list) for having a stable and converged model. In the CFD analysis, the 4-node FCBI finite element has been used.

## **Key Words**

Clark Y foil, Kutta condition, CFD, flow-condition-based interpolation



Department of Mechanical Engineering  
Massachusetts Institute of Technology

2.094 - Term Project - Hydrofoil analysis using CFD

Ofer Aharon

## Table of Contents

List of Figures .....	iii
List of Tables .....	iii
1.0 Title .....	1
2.0 Nomenclature .....	1
3.0 Introduction .....	2
4.0 Theory of Wing Sections .....	3
5.0 The geometric model .....	6
6.0 CFD Analysis of one foil .....	11
7.0 CFD Analysis of two sequential foils .....	18
8.0 Summary .....	20
9.0 References .....	21



### List of Figures

Figure 1. Infinite velocity at trailing edge. .... 3  
 Figure 2. Circulatory flow only. .... 3  
 Figure 3. Uniform stream plus Circulatory..... 4  
 Figure 4. Transformation from a circle to noncircular curve into a foil. .... 5  
 Figure 5. Pressure distribution along the foil using [4] points..... 7  
 Figure 6. Pressure distribution along the foil using points from [4]. .... 7  
 Figure 7. Correct model of Clark-Y foil using Unigraphics..... 10  
 Figure 8. The meshed model..... 12  
 Figure 9. The CFD solution for one foil. .... 14  
 Figure 10. Comparison of pressure distribution with  $\alpha= 9.55^\circ$ . .... 15  
 Figure 11. Comparison of pressure distribution with  $\alpha= 5.32^\circ$ . .... 15  
 Figure 12. Comparison of pressure distribution with  $\alpha= -1.27^\circ$ . .... 16  
 Figure 13. Comparison of pressure distribution with  $\alpha= -3.25^\circ$ . .... 16  
 Figure 14. The geometry description of the two sequential foils. .... 18  
 Figure 15. The CFD solution for two sequential foils. .... 18  
 Figure 16. Comparison of pressure distribution between two sequential foils..... 19

### List of Tables

Table 1. Clark-Y foil coordinates from [2]..... 6  
 Table 2. Detailed Clark-Y foil coordinates using Unigraphics - upper surface..... 8  
 Table 3. Detailed Clark-Y foil coordinates using Unigraphics - lower surface..... 9



Department of Mechanical Engineering  
Massachusetts Institute of Technology

2.094 - Term Project - Hydrofoil analysis using CFD

Ofer Aharon

## 1.0 Title

2.094 Term Project - Hydrofoil analysis using CFD.

## 2.0 Nomenclature

$U_\infty$  - Uniform stream velocity [m/sec].

$v$  - Velocity on the foil [m/sec].

$p_\infty$  - Pressure far away from the foil [Pa].

$p_f$  - Pressure on the foil [Pa].

$p$  - Relative pressure on the foil [Pa].

$\Gamma$  - Circulation [ $\text{m}^2/\text{sec}$ ].

$\rho$  - Density of the fluid [ $\text{kg}/\text{m}^3$ ].

$x$  - Dimensionless x-coordinate of the foil.

$y_U$  - Dimensionless y-coordinate of the upper surface of the foil.

$y_L$  - Dimensionless y-coordinate of the lower surface of the foil.

$C_p$  - Dimensionless pressure.

$\alpha$  - Angle of attack of the foil [deg].



### **3.0 Introduction**

Solving for a pressure field around an arbitrary body in a flow field is very meaningful for many physical applications. Many engineering issues consist of flow around a body, for example: flying airplane, ship at seaway, wind influence on structures, flow in veins etc. Thus, it is most important to find the velocities and pressures fields on a body in as short time as possible with adequate resources.

Since it is complicated and some of the time impossible to find an analytical solution but still essential, the Finite Elements Method (FEM) has been taken place. This method, in general, divides the control volume to many small (but finite) elements where the shared points of the elements are called nodes. The algorithm runs over the nodes till equilibrium is achieved in every node. The FEM is called Computational Fluid Dynamics (CFD) when a flow field is involved. The CFD method is very expensive in time and resources since a solution must be achieved all over the control volume of the problem and not just around the body itself. Thus, the mesh of the control volume becomes very sophisticated and has many elements. Consequently, even when high-speed [supercomputers](#) are used, it takes a few days to get a solution. It should be notify that the gained solution is only an accurate one but not an exact one.

The goal of this project is to find the distributed pressure around two following foils where the distance between them equals their chord length. In order to verify the process and the results, a comparison was made between the distributed pressure curves around Clark Y foil of analytical, experimental and ADINA results.



### 4.0 Theory of Wing Sections

Knowing the pressure field around a wing is of vital importance in hydrodynamics. In order to find the distributed pressure along the upper and lower surfaces of the wing, a uniform stream is imposed. According [1]<sup>1</sup>, applying only uniform stream will result in infinite velocity at the trailing edge like described in the next figure (streamlines past a wing):

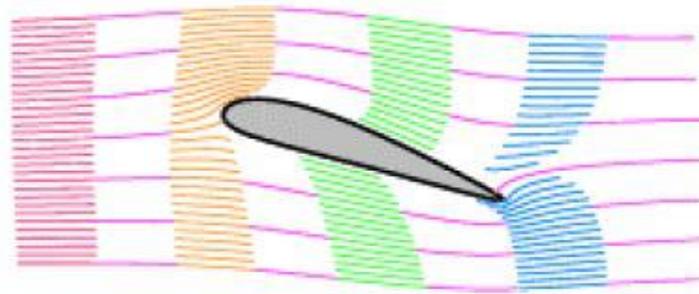


Figure 1. Infinite velocity at trailing edge.

This infinite velocity violates Kutta condition which states that **“flow leaves tangentially the trailing edge, i.e., the velocity at the trailing edge is finite”**. In order to satisfy Kutta condition, circulation is applied. The next figure describes circulatory flow:

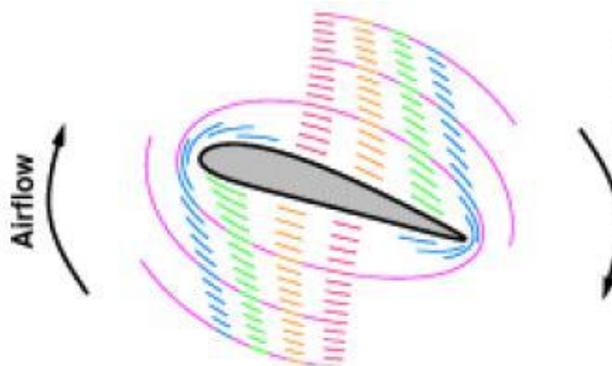


Figure 2. Circulatory flow only.

<sup>1</sup> Numbers in rectangular parenthesis represent the reference number in the references list.



Consequently, the flow is combined from uniform stream and circulation:

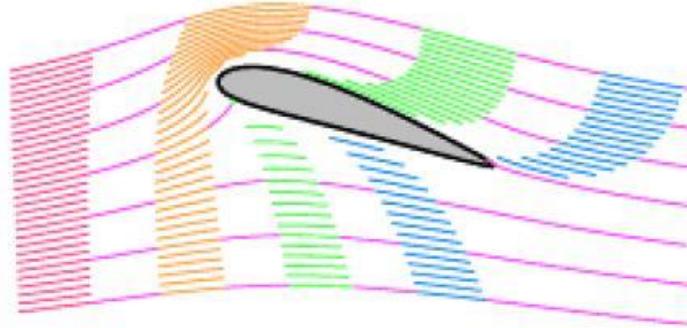


Figure 3. Uniform stream plus Circulatory.

When a wing starts moving, vortices are created behind due to separation at the tail (infinite velocity). The vortices shedding continues till Kutta condition is satisfied and the flow leaves the wing smoothly. Thus, a potential flow is considered when solving for pressure field around a wing. For streamlined asymmetric body, under small angles of attack, there is no separation even for large Reynolds numbers. In addition, the viscous effects are only in a thin boundary layer, thus, the drag on the wing is very small (only skin friction) and can be neglected when the fluid is sea water ( $\nu \approx 10^{-6} \frac{m^2}{sec}$ ).

The velocity potential  $\Phi$  becomes:

$$(1) \quad \Phi = U_{\infty} \cdot x + \frac{\Gamma}{2\pi} \cdot \tan^{-1} \left( \frac{y - y_0}{x - x_0} \right) + C$$

Solving a potential flow around a circle with radius 'a' is a simple and straight forward problem. Using mathematical transformation between circle and wing, one can find the pressure along any arbitrary wing when the above conditions are preserved. Such a process is described in details in [2] and the next figure explains the transformation:

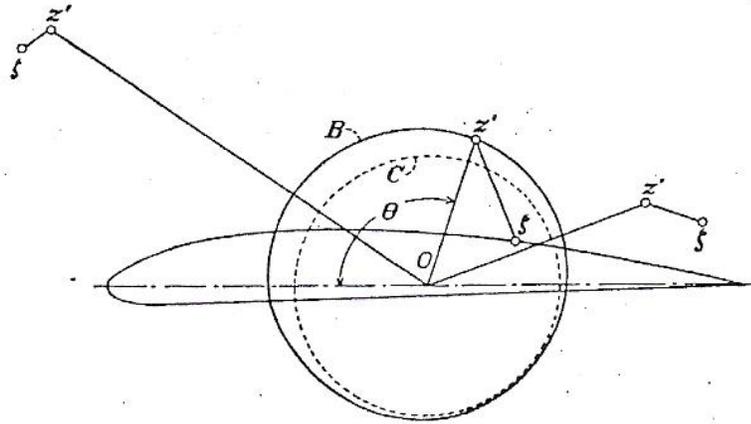


Figure 4. Transformation from a circle to noncircular curve into a foil.

The analytical solution given in [2] is sophisticated and involves De Moivre's theorem with Fourier expansions in addition to the transformations of coordinates. Since the mathematical process is too long to describe, it is omitted here in this term project. Thus, only the final results of the analytical solution are given here.



### 5.0 The geometric model

The wing section that was analyzed in [2] is the Clark-Y foil where its coordinates are given in [2]:

Table 1. Clark-Y foil coordinates from [2].

%c	x	yU	yL
0	0	0.000	0.000
1.25	0.05	0.0803	-0.0618
2.5	0.101	0.124	-0.0787
5	0.202	0.185	-0.0961
7.5	0.303	0.226	-0.105
10	0.404	0.260	-0.110
15	0.605	0.311	-0.115
20	0.807	0.345	-0.112
30	1.211	0.373	-0.0997
40	1.614	0.375	-0.0856
50	2.0175	0.353	-0.0718
60	2.421	0.314	-0.0557
70	2.825	0.255	-0.0416
80	3.228	0.181	-0.0299
90	3.631	0.097	-0.0161
95	3.833	0.050	-0.0097
100	4.036	0.002	-0.002

However, a much more detailed leading edge is necessary for CFD analysis. Reference [3] also doesn't give enough points so consequently, looking for detailed Clark-Y section, I've found the Airfoil Design Workshop software [4] that had 142 points of the section. When trying to solve for the field flow around the foil, one would get the next results:

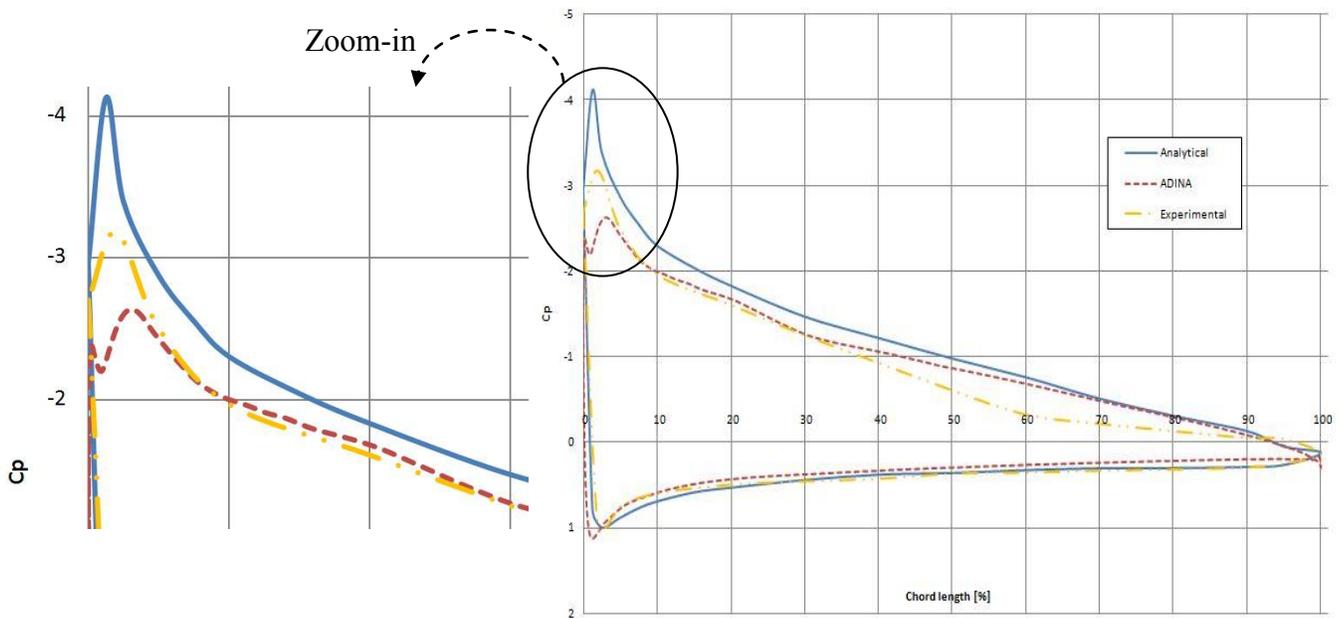


Figure 5. Pressure distribution along the foil using [4] points.

One can see that at the leading edge, the results are not physical at all. The pressure curve isn't smooth and since the flow is very sensitive to the curvature of the foil, the points of the section were investigated using Unigraphics and XL:

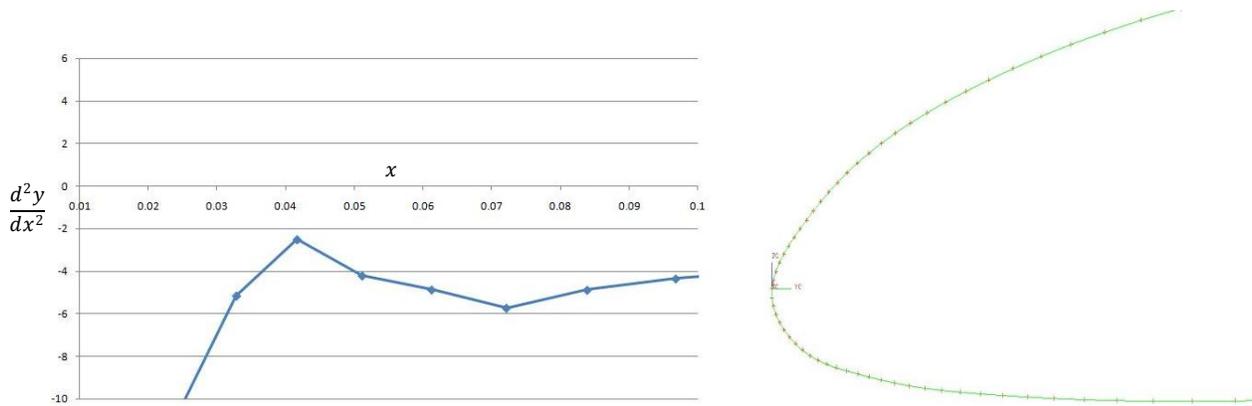


Figure 6. Pressure distribution along the foil using points from [4].

From the graph in XL it is seen that the curvature is approaching zero but at 4% of the chord length it is increasing till 7% of the chord length and then decreasing again. In order to achieve the correct curve with a monotonic and smooth curvature (2<sup>nd</sup> derivative), I've used Unigraphics and the points of the section are given in the next tables:



Department of Mechanical Engineering  
Massachusetts Institute of Technology

2.094 - Term Project - Hydrofoil analysis using CFD

Ofer Aharon

Table 2. Detailed Clark-Y foil coordinates using Unigraphics - upper surface.

<b>x</b>	<b>y<sub>U</sub></b>	<b>x</b>	<b>y<sub>U</sub></b>	<b>x</b>	<b>y<sub>U</sub></b>
0	0	0.80680	0.33090	3.56193	0.10998
0.00085	0.01261	0.88090	0.33965	3.64858	0.09133
0.00210	0.01983	0.95794	0.34744	3.73447	0.07241
0.00589	0.03321	1.03757	0.35417	3.81927	0.05323
0.01110	0.04554	1.11947	0.35979	3.90249	0.03394
0.01744	0.05701	1.20313	0.36423	3.98357	0.01485
0.02470	0.06777	1.28813	0.36743	4.036	0.00242
0.03277	0.07795	1.37389	0.36936		
0.04161	0.08771	1.46006	0.37000		
0.05110	0.09703	1.54643	0.36921		
0.06123	0.10602	1.63301	0.36748		
0.07212	0.11485	1.71974	0.36457		
0.08391	0.12363	1.80668	0.36054		
0.09682	0.13250	1.89385	0.35545		
0.11099	0.14150	1.98119	0.34944		
0.12649	0.15064	2.06869	0.34245		
0.14352	0.15998	2.15631	0.33467		
0.16221	0.16950	2.24394	0.32599		
0.18267	0.17920	2.33164	0.31646		
0.20515	0.18913	2.41938	0.30605		
0.22993	0.19931	2.50716	0.29483		
0.25725	0.20975	2.59503	0.28276		
0.28744	0.22045	2.68301	0.27001		
0.32078	0.23141	2.77108	0.25653		
0.35759	0.24259	2.85922	0.24244		
0.39827	0.25397	2.94737	0.22775		
0.44319	0.26552	3.03552	0.21254		
0.49263	0.27713	3.12362	0.19671		
0.54676	0.28868	3.21165	0.18037		
0.60552	0.30001	3.29955	0.16350		
0.66872	0.31095	3.38729	0.14610		
0.73596	0.32130	3.47479	0.12826		



Department of Mechanical Engineering  
Massachusetts Institute of Technology

2.094 - Term Project - Hydrofoil analysis using CFD

Ofer Aharon

Table 3. Detailed Clark-Y foil coordinates using Unigraphics - lower surface.

X	y <sub>L</sub>	X	y <sub>L</sub>	X	y <sub>L</sub>
0	0	0.82577	-0.11930	3.57150	-0.01949
0.00004	-0.00965	0.90140	-0.11725	3.65690	-0.01635
0.00202	-0.01885	0.97998	-0.11470	3.74141	-0.01324
0.00597	-0.02793	1.06094	-0.11184	3.82463	-0.01017
0.01146	-0.03657	1.14372	-0.10873	3.90612	-0.00718
0.01824	-0.04480	1.22783	-0.10554	3.98507	-0.00428
0.02615	-0.05255	1.31287	-0.10235	4.036	-0.00242
0.03511	-0.05981	1.39860	-0.09920		
0.04508	-0.06655	1.48476	-0.09606		
0.05610	-0.07249	1.57130	-0.09291		
0.06825	-0.07765	1.65799	-0.08976		
0.08141	-0.08201	1.74488	-0.08657		
0.09557	-0.08581	1.83182	-0.08338		
0.11071	-0.08911	1.91888	-0.08020		
0.12681	-0.09238	2.00593	-0.07697		
0.14404	-0.09577	2.09307	-0.07378		
0.16241	-0.09912	2.18017	-0.07059		
0.18219	-0.10235	2.26734	-0.06736		
0.20350	-0.10534	2.35448	-0.06417		
0.22658	-0.10800	2.44166	-0.06098		
0.25168	-0.11030	2.52880	-0.05776		
0.27905	-0.11232	2.61597	-0.05457		
0.30904	-0.11414	2.70311	-0.05138		
0.34197	-0.11583	2.79029	-0.04819		
0.37817	-0.11753	2.87743	-0.04496		
0.41813	-0.11910	2.96452	-0.04177		
0.46220	-0.12043	3.05162	-0.03858		
0.51080	-0.12144	3.13864	-0.03540		
0.56423	-0.12205	3.22561	-0.03217		
0.62259	-0.12221	3.31243	-0.02898		
0.68584	-0.12181	3.39912	-0.02583		
0.75372	-0.12084	3.48549	-0.02264		



The next figures represent the foil as modeled in Unigraphics:

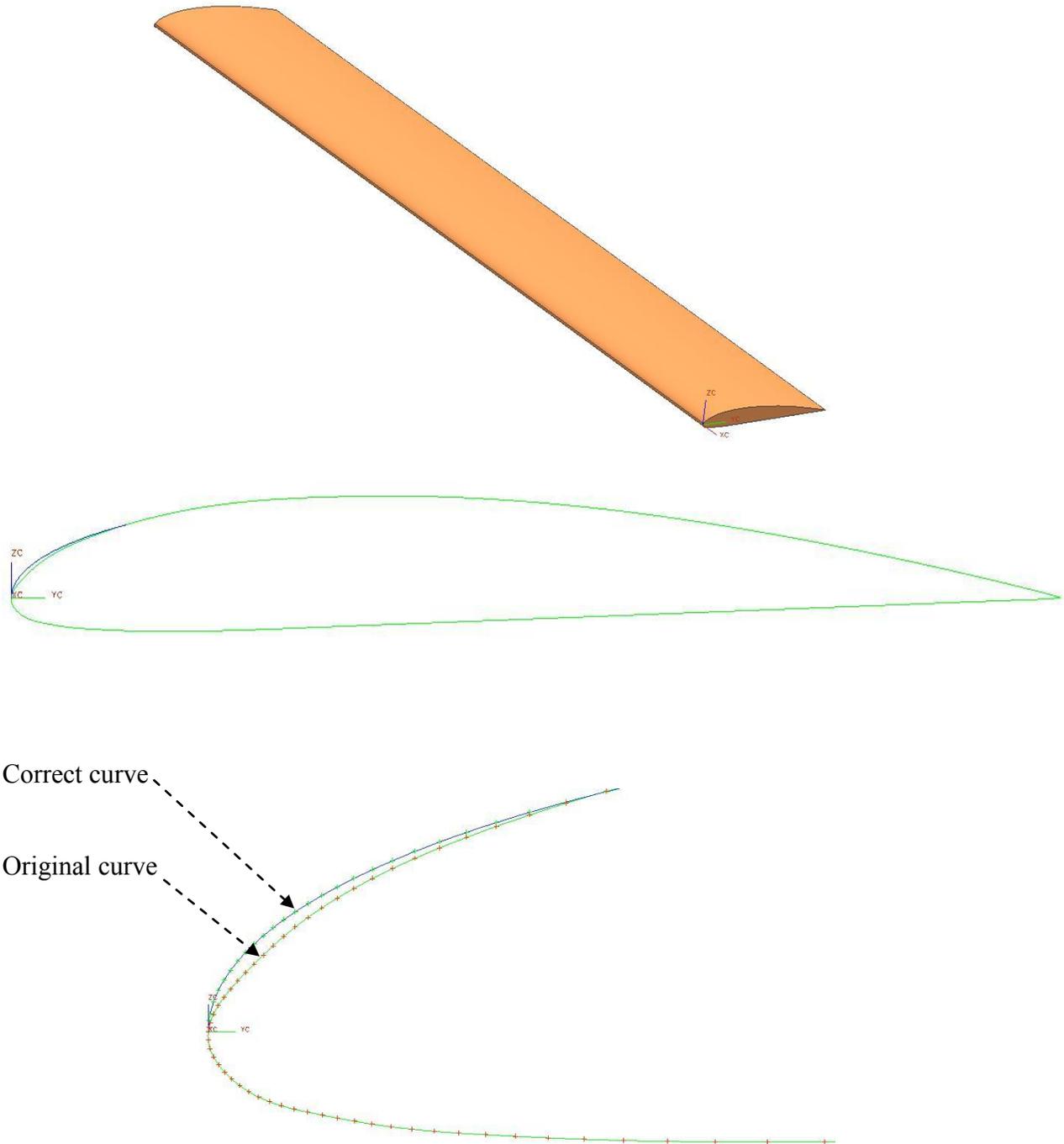


Figure 7. Correct model of Clark-Y foil using Unigraphics.



## **6.0 CFD Analysis of one foil**

The CFD analysis was performed by using the Automatic Dynamic Incremental Nonlinear Analysis (ADINA) software. ADINA R & D, Inc. was founded in 1986 by Prof. [K. J. Bathe](#) and associates. The exclusive mission of the company is the development of the ADINA System for the analysis of solids, structures, fluids and fluid flow with structural interactions.

For the given geometry, 4 different angles of attack have been considered in [2]:  $+9.55^\circ$ ,  $+5.32^\circ$ ,  $-1.27^\circ$ ,  $-3.25^\circ$ . All of them were analyzed analytically and the results of the distributed pressure were approved in experiments. Those same angles of attack were examined in ADINA and the accuracy of results is very high as detailed in the following.

The presented problem is a 2D sea water flow around the foil. The fluid density is  $1025 \text{ kg/m}^3$  and its dynamic viscosity is neglected. The solution we are after is dimensionless pressure distribution ( $C_p$ ) along the foil which is derived by the Bernoulli equation for a steady-state flow:

$$(2) \quad p_f + \frac{1}{2}\rho \cdot v^2 = p_\infty + \frac{1}{2}\rho \cdot U_\infty^2$$

$$(3) \quad p_f - p_\infty + \frac{1}{2}\rho \cdot v^2 = \frac{1}{2}\rho \cdot U_\infty^2 \equiv q$$

$$(4) \quad p = p_f - p_\infty$$

$$(5) \quad \frac{p}{q} + \frac{v^2}{U_\infty^2} = 1$$

$$(6) \quad C_p = \frac{p}{q} = 1 - \frac{v^2}{U_\infty^2}$$



One should notice that, although a potential flow is solved by the Laplace equation in 2D which applies also to heat transfer, one **cannot** use ADINA-T for solving the flow around the foil. The reason for that is that vorticity vanishes in this case like it is stated in [5]:

$$(7) \quad \frac{\partial v_x}{\partial y} - \frac{\partial v_y}{\partial x} = 0$$

Since vorticity vanishes, the Kutta condition will not be satisfied as explained above. Thus, ADINA-F was used for this model. Meshing the control surface around the foil and applying the boundary conditions of free slip on the foil leads to the following meshed model:

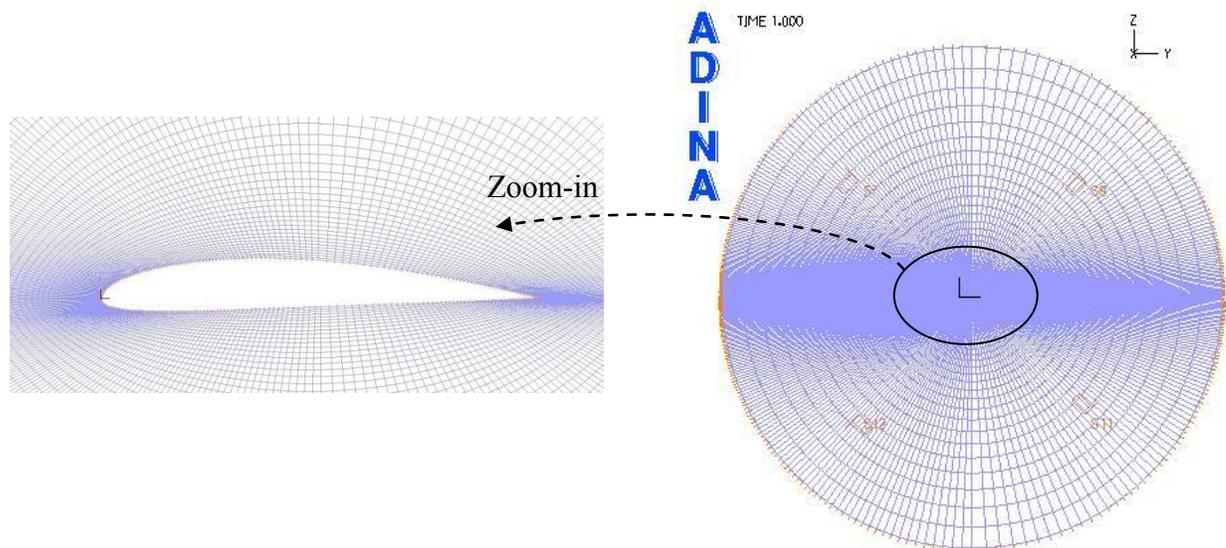


Figure 8. The meshed model.

The CFD procedures are more sophisticated than those of structures methods. The 3D structures model is the structure itself with the loads applied on it. On the other hand, the entire environment should be meshed and not just the body itself when solving a 2D flow problem. Thus, the number of elements of the meshed model is 50 times than the number of a structural meshed model. In order to solve a 2D flow problem, it takes time and special resources. In order to solve a 3D flow problem, it gets away too complicated. Thus, a lot of effort is invested in

	<b>Department of Mechanical Engineering</b> <b>Massachusetts Institute of Technology</b>	
	2.094 - Term Project - Hydrofoil analysis using CFD	Ofer Aharon

looking for procedures and element-types that are stable and give accurate solution but the time consuming is reduced significantly.

In order to reduce the time of a process, the algorithm of solving the CFD equations has to be simplified. One of the approaches is to change the interpolation procedure of the equations in order to get more stable and accurate solution in each node of the meshed model. According [6], there are many techniques for solving fluid flow problems. A very important and developed procedure is the flow-condition-based interpolation (FCBI). This interpolation has two meaningful aspects which are stability and flux equilibrium.

Another approach is using a higher-order finite element with more nodes. This will promise a more accurate solution. A formulation of 9-node FCBI finite element is presented in [7]. According this article, an effective fluid flow analysis was obtained in several cases using coarse meshes. Like in structures analyses, the 9-node element is a very good candidate for error assessment although it takes more time for the computer to solve the model. The article states that the element was examined for 2D flow, but may be employed to 3D analyses as well.

A study of a 2D flow, where the cubic interpolated polynomial (CIP) method is used with 4-node and 9-node finite elements, is detailed in [8]. The reason for using the CIP method is to stabilize the convective terms of the CFD equations. The goal is to make a coarse mesh of the model in order to save time of solving the equations but still to get an accurate solution. According [8], the 4-node and 9-node elements show accurate and stable solutions, but were too expensive in the applications. It took too much time to get the solution than expected. In addition, the authors state that the article results are based on numerical procedures and not on mathematical-analytical ones.

Consequently, in order to reduce the time of the solution and still get stable and converged model, I've used the 4-node FCBI finite element in the model.



Solving the model for the 4 angles of attack, one gets the following solution. The upper part of each figure is the velocity profile and the lower one is the pressure field around the foil:

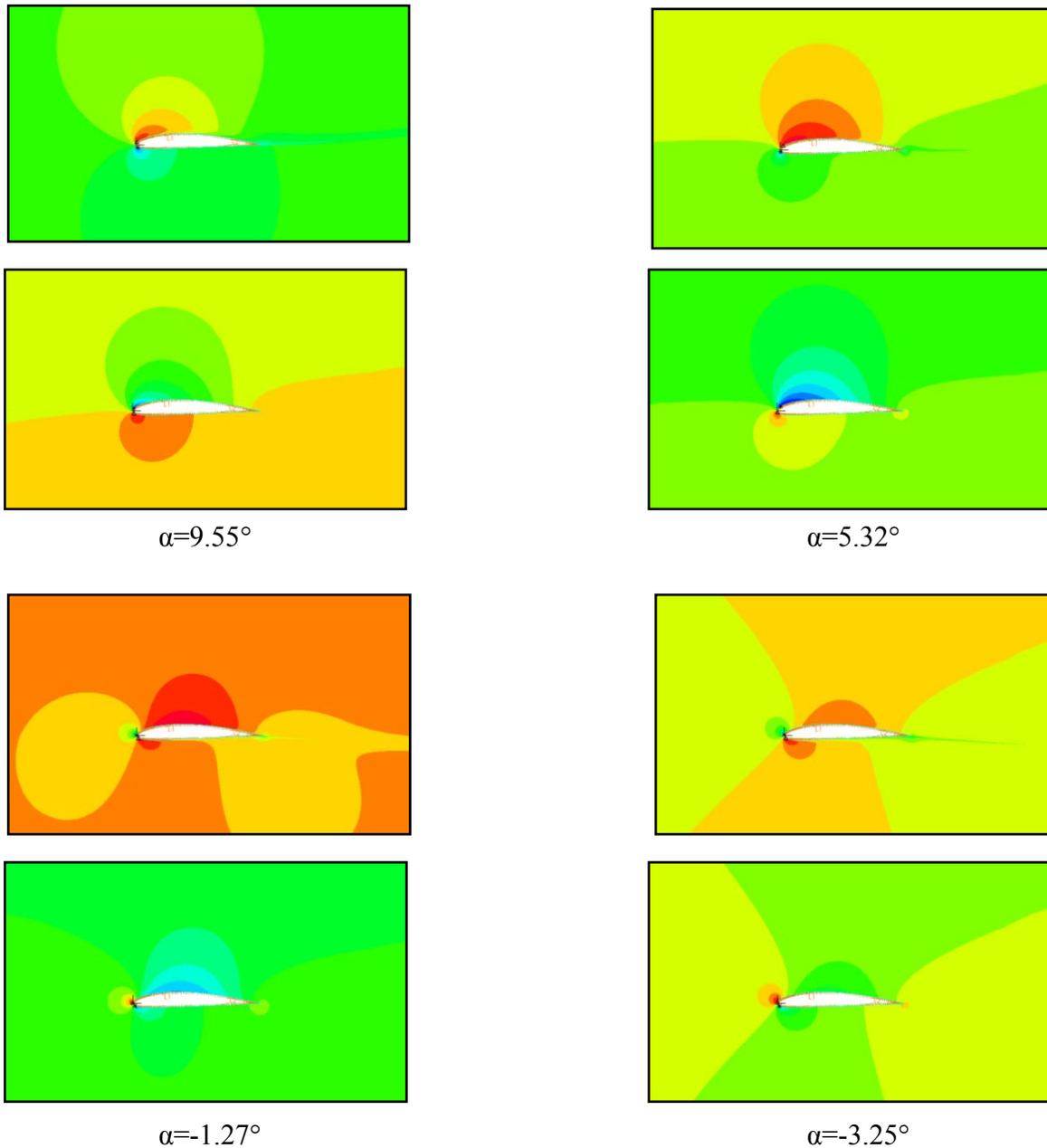


Figure 9. The CFD solution for one foil.



The next figures compare between the analytical solution, experiments and the CFD results. Each figure shows the pressure distribution along the upper and lower surface of the foil. The graphs are all dimensionless when the y-coordinate is the dimensionless pressure ( $C_p$ ) and the x-coordinate is the length chord in %.

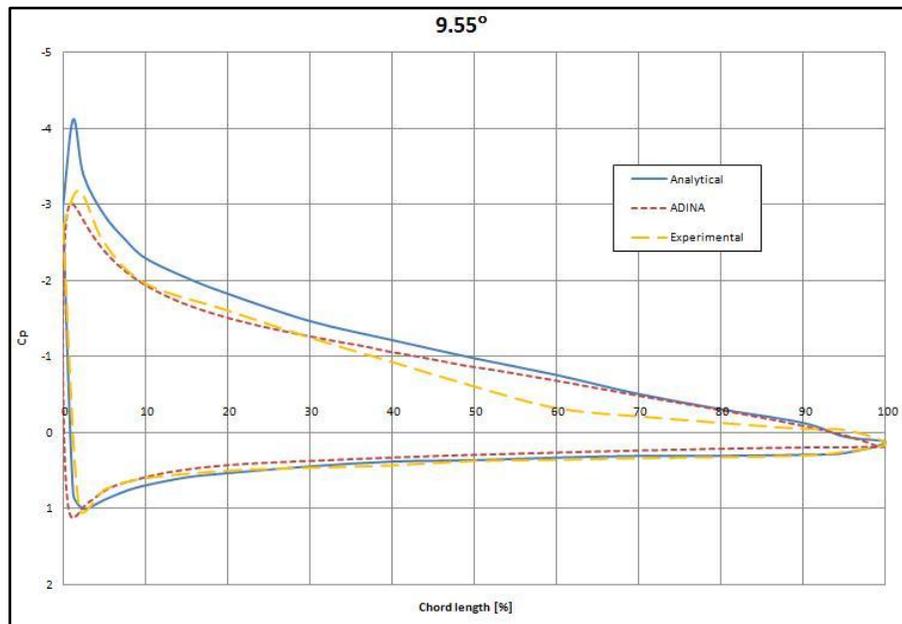


Figure 10. Comparison of pressure distribution with  $\alpha= 9.55^\circ$ .

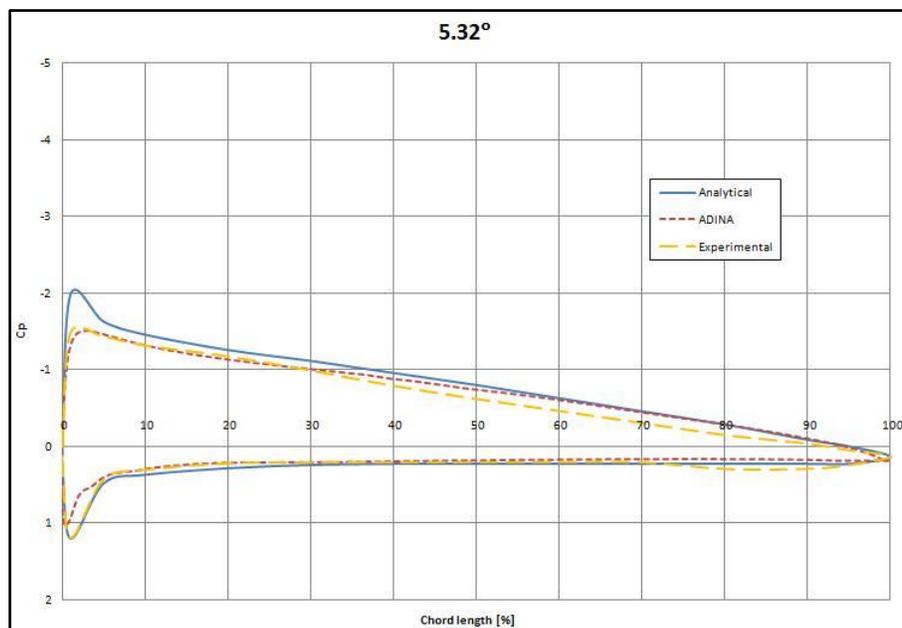


Figure 11. Comparison of pressure distribution with  $\alpha= 5.32^\circ$ .

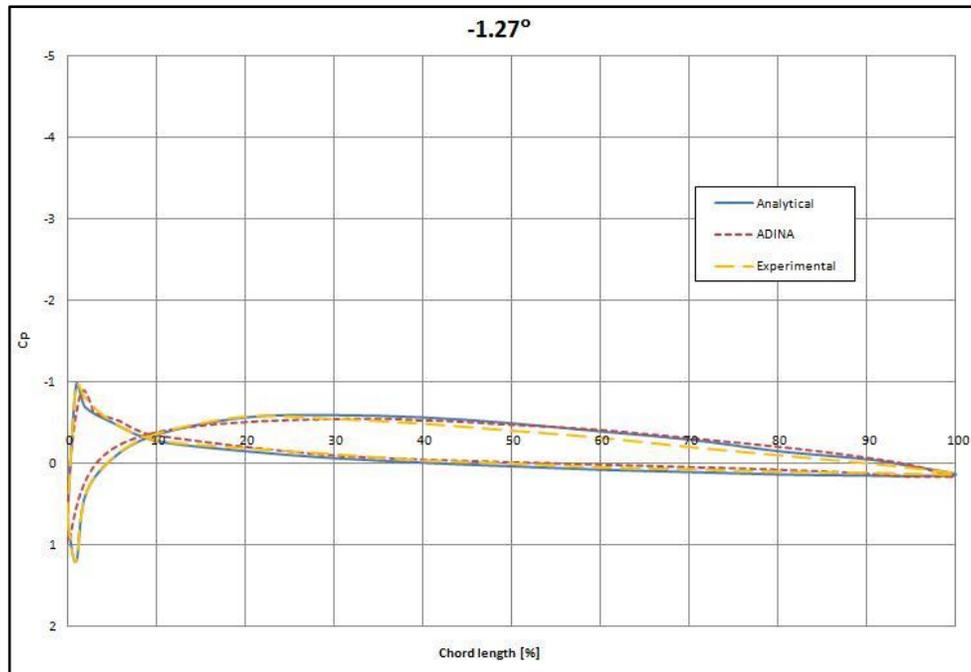


Figure 12. Comparison of pressure distribution with  $\alpha = -1.27^\circ$ .

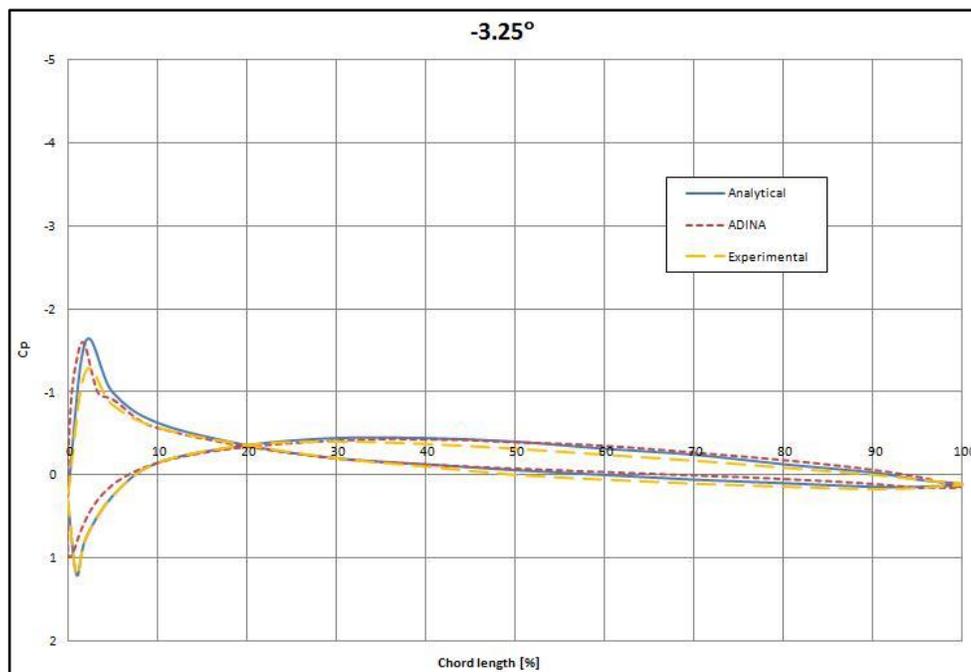


Figure 13. Comparison of pressure distribution with  $\alpha = -3.25^\circ$ .



## Department of Mechanical Engineering Massachusetts Institute of Technology

2.094 - Term Project - Hydrofoil analysis using CFD

Ofer Aharon

It can be seen that the pressure distribution along the curve is almost the same for each one of the four angles of attack. There could be many reasons for the slight differences such as:

1. The analytical solution is based on the mentioned assumptions where are close to reality but not exact.
2. The analytical solution is solving the mathematical model and not the physical model.
3. The foil in the experiments wasn't smooth and had some humps along it.
4. The measurements at the experiments weren't exact and also the adjustment of the angle of attack of the foil.
5. The viscosity of the fluid isn't zero and there are boundary layers effects that needed to be taken into account.
6. Small separations of the flow might have taken place right after the leading edge.

In summary, one can see that the accuracy of the results is very high and definitely satisfactory.



### 7.0 CFD Analysis of two sequential foils

After solving the flow around the foil using ADINA and successfully comparing it with given analytical and experimental data, the next stage was to examine the influence of two same foils at zero angle of attack when the horizontal distance between them is their chord length.

The next figure describes the geometry of the problem:

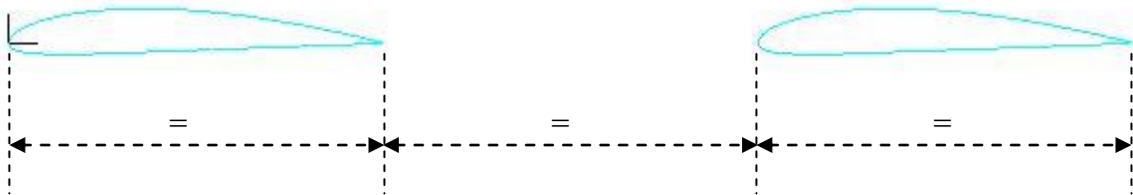


Figure 14. The geometry description of the two sequential foils.

The following solution is achieved when solving this model for zero angle of attack. The upper part of the figure is the velocity profile and the lower one is the pressure field around the foils:

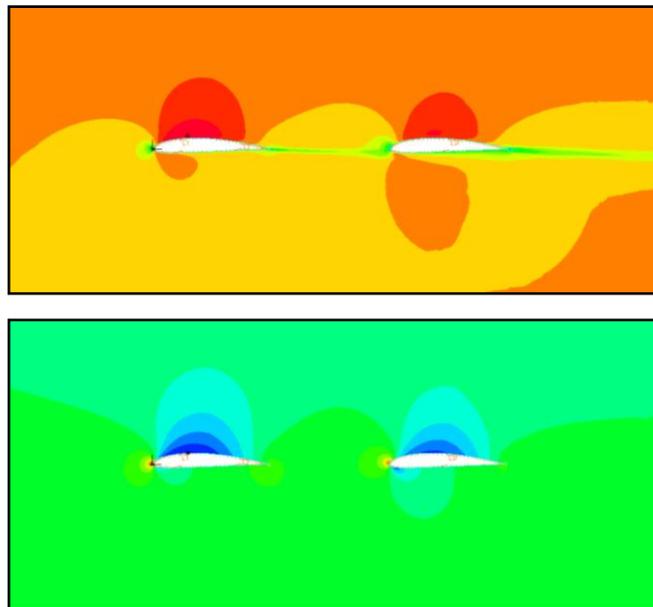


Figure 15. The CFD solution for two sequential foils.



The next figure compares between the pressure distribution along the first foil and the second foil. The graph is dimensionless when the y-coordinate is the dimensionless pressure ( $C_p$ ) and the x-coordinate is the length chord in %.

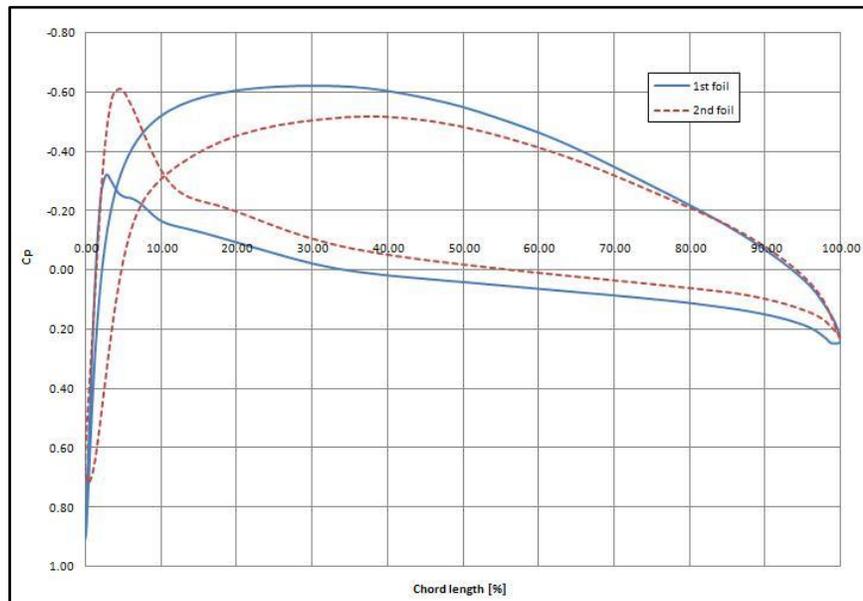


Figure 16. Comparison of pressure distribution between two sequential foils.

Several conclusions can be made by observing this graph:

1. The  $C_p$  value of the 2<sup>nd</sup> foil is lower than of the 1<sup>st</sup> foil in most of the chord length.
2. The confined area of the distributed pressure is the lift gained by the foil. It can be seen that the confined area of the 1<sup>st</sup> foil is bigger than the confined area of the 2<sup>nd</sup> foil; that means that the lift of the 1<sup>st</sup> foil is bigger.
3. Although the difference is only one chord length between the two foils, the fluid flow around the 2<sup>nd</sup> foil is not interrupted in a way which the 2<sup>nd</sup> foil doesn't contribute sufficient lift relative to the 1<sup>st</sup> foil which is not interrupted.
4. The curve of the 2<sup>nd</sup> foil is similar to a curve of one foil under little negative angle of attack (See Fig. 12). The 2<sup>nd</sup> foil experiences such an angle since the flow leaves the first foil with positive angle which represents the slope of foil at the tail (Kutta condition is satisfied and the flow leaves the 1<sup>st</sup> foil tangentially).

	Department of Mechanical Engineering Massachusetts Institute of Technology	
	2.094 - Term Project - Hydrofoil analysis using CFD	Ofer Aharon

## 8.0 Summary

The goal of this project is to find the influence of two sequential foils on each other when are very close horizontally. Since it is very complicated to find an analytical solution, I've used a Finite Element software called ADINA.

In order to have confidence in the process and to verify the solution, I've solved an analytical solution for a Clark Y foil under 4 different angles of attack. The full detailed solution is given in [2] and since it is too long, it is omitted from this project. In addition, there were experimental results given with the analytical solution. Both the analytical and experimental results appear above.

When solving the same foil in ADINA, I've found out that the offsets of the foil given in the literature aren't detailed enough and when I've taken new offsets (142 points) from another program (XFOIL), it was found out as problematic and not physical at the leading edge. Thus, I've used the Unigraphics software to correct the leading edge by having a smooth spline with continuous and smooth curvature. The new and correct offsets of the Clark Y foil are given in a table. The modeled foil in Unigraphics is also presented above.

Solving for the pressure distribution along the foil and comparing it with the analytical and experimental results, one can find the impressive accuracy of the curves. The slight differences between the curves are due to the assumptions of the analytical solution and the non-ideal conditions of the experiments as detailed above.

After verifying the solution of the flow with a given analytical and experimental data, I've moved to the solution of two sequential wings where both are in the same length and same angle of attack ( $0^\circ$ ). The horizontal distance between them was their chord length. This solution was done by using ADINA. The results of this problem are detailed above. Comparing the pressure distributions of the two foils, one can concludes that the results make sense and are very physical.

In addition, some literature was reviewed (see next page for references list) when looking for a stable and converged model. Many ways are being searched for reducing the time of the process and getting a more accurate solution. The main efforts focus on using different algorithms and interpolations methods, and also higher-order finite element for meshing the model. A very important and developed procedure is the flow-condition-based interpolation (FCBI) which leads to stability and flux equilibrium of the model. Thus, the 4-node FCBI finite element has been used in the above CFD analyses.



## 9.0 References<sup>2</sup>

- [1] J.N. Newman, *Marine Hydrodynamics*, MIT, 1978.
- [2] Theodore Theodorsen. *Theory of wing sections of arbitrary shape*. NACA Report No.411, 1932.
- [3] Riegels, F.W., *Airfoil Sections*, Butterworths, London, 1961 (English translation from German).
- [4] Airfoil Design Workshop, copyright 2003-2006, [www.tdmssoftware.com](http://www.tdmssoftware.com).
- [5] Prof. Bathe KJ., *Finite Element Procedures*, Prentice Hall, NJ, 1996.
- [6] Kohno H, Prof. Bathe KJ. *Insight into the flow-condition-based interpolation finite element approach: solution of steady-state advection-diffusion problems*. International Journal for Numerical Methods in Engineering 2005; **63**:197-217.
- [7] Kohno H, Prof. Bathe KJ. *A nine-node quadrilateral FCBI element for incompressible fluid flows*. Communications in Numerical Methods in Engineering 2006; **22**:917-931.
- [8] Banijamali B, Prof. Bathe KJ. *The CIP method embedded in finite element discretizations of incompressible fluid flows*. International Journal for Numerical Methods in Engineering 2007; **71**:66-80.

---

<sup>2</sup> This list is sorted by the order of the reference appearance in this document.

MIT OpenCourseWare  
<http://ocw.mit.edu>

2.094 Finite Element Analysis of Solids and Fluids  
Spring 2011

For information about citing these materials or our Terms of Use, visit: <http://ocw.mit.edu/terms>.