Flow Over a ClarkY Airfoil

Abstract—*In this exercise, flow over a ClarkY airfoil is modeled. The flow is assumed to be two-dimensional. The chord length of the airfoil is 0.4036 m. Coarse, medium, and fine mesh types are available. Inviscid or viscous flow fields can be simulated over the airfoil. The material properties of air are used with ideal gas behavior assumed for density calculations. Far-field temperature, far-field pressure, Mach number, and angle of attack can be specified as the boundary conditions. The exercise reports wall shear stress, skin friction factor, lift coefficient, and drag coefficient. Plots of pressure coefficient, friction coefficient, and shear stress distribution are available. Contours of pressure, velocity, temperature, Mach number, stream function, turbulent kinetic energy, and dissipation rate can be displayed. A velocity vector plot is also available.*

1 Introduction

A fundamental and common aerodynamics problem of practical importance is the flow over an airfoil. Airfoils are two-dimensional representations of three-dimensional wings. Two-dimensional airfoils can be considered the building blocks of a wing, meaning the aerodynamic characteristics of airfoils are useful for interpreting the performance of the whole wing. In this exercise, both inviscid and viscous flows over an airfoil are investigated. The lift and drag characteristics of the ClarkY airfoil and the variation of lift with the angle of attack can be evaluated. You can also compare pressure distribution for viscous and inviscid flow fields, so that the effects of viscosity can be interpreted.

2 Modeling Details

The fluid region is represented in two dimensions. The procedure for solving the problem is:

1. Create the geometry.
2. Set the material properties and boundary conditions.
3. Mesh the domain.

FlowLab creates the geometry and mesh, and exports the mesh to FLUENT. The boundary conditions and flow properties are set through parameterized case files. FLUENT converges the problem until the convergence limit is met or the number of iterations specified is achieved.

2.1 Geometry

The ClarkY airfoil is created from a set of vertices. These vertices are connected using a smooth curve, creating the surface of the airfoil. A flow domain is created surrounding the airfoil and this domain is split for meshing purposes (Figure 2.1). The following dimensions (in SI) are used for the domain:

- W = 10 m
- L = 12.5 m
- C = 0.4036 m (where C is the chord length of the airfoil)

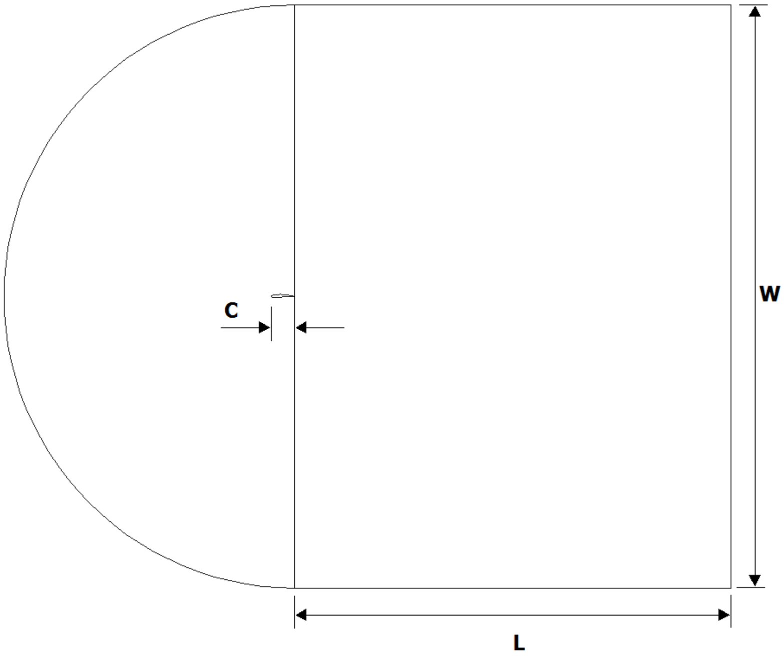

Figure 2.1: Schematic of the Flow Domain

2.2 Mesh

Coarse, medium, and fine mesh types are available. Mesh density varies based upon the assigned Refinement Factor. The Refinement Factor values for the mesh densities are given in Table 2.1.

Mesh Density	**Refinement Factor**
Fine	1
Medium	1.5
Coarse	2.25

Table 2.1: Refinement Factor

Using the Refinement Factor, First Cell Height is calculated with the following formula:

$$First\ Cell\ Height = Refinement\ Factor \left[\frac{Yplus \times (Characteristic\ Length^{0.125} \times Viscosity^{0.875})}{(0.199 \times Velocity^{0.875} \times Density^{0.875})} \right] \tag{2-1}$$

Reynolds number based upon chord length is used to determine Yplus. Yplus values for turbulent flow conditions are summarized in Table 2.2.

Reynolds Number	**Flow Regime**	**Yplus/First Cell Height**
Re $\leq$ 50000	Turbulent, Enhanced Wall Treatment	Yplus $<$ 10
Re $>$ 50000	Turbulent, Standard Wall Functions	Yplus $>$ 30

Table 2.2: Flow Regime Vs. Reynolds Number

The number of intervals along each edge is determined using geometric progression and the following equation:

$$Intervals = INT\left[\frac{Log\left\{\frac{Edge\_length \times (Growth\_ratio-1)}{First\ Cell\ Height} + 1.0\right\}}{Log(Growth\_ratio)}\right] \tag{2-2}$$

The edges are meshed using the First Cell Height and the calculated number of intervals. The entire domain is meshed using a map scheme. The resulting mesh is shown in Figure 2.2.

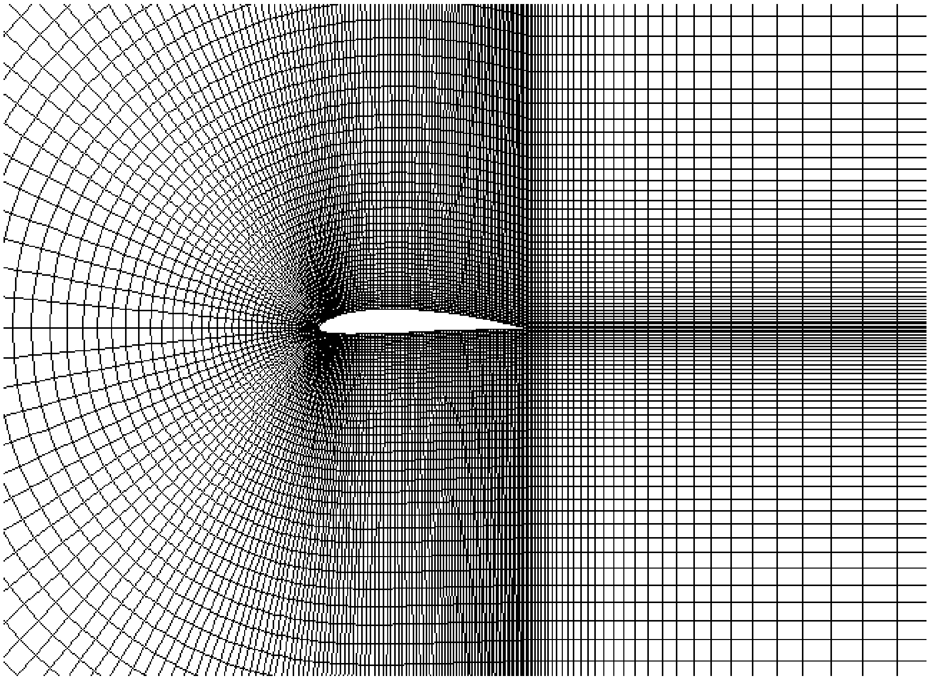

Figure 2.2: Mesh Generated by FlowLab

2.3 Physical Models for FLUENT

Either inviscid flow or viscous flow models can be applied. Based on the Reynolds number, the following physical models are applied:

Re $\leq$ 50000	$k - \epsilon$ model
Re $>$ 50000	$k - \epsilon$ model

Table 2.3: Turbulence Models Based on Reynolds Number

If turbulence is selected in the Physics form of the Operation menu, the appropriate turbulence model and wall treatment is applied based upon the Reynolds number.

2.4 Material Properties

The following material properties of the fluid can be specified:

- Density (ideal gas law)
- Thermal conductivity (only for viscous flow)
- Viscosity (only for viscous flow)
- Specific heat
- Molecular weight

2.5 Boundary Conditions

You can specify the following boundary conditions:

- Far field pressure
- Far field temperature
- Mach number
- Angle of attack

The following boundary conditions are assigned in FLUENT.

Boundary	**Assigned As**
Airfoil	Wall
External domain	Pressure far-field

Table 2.4: Boundary Conditions Assigned in FLUENT

2.6 Solution

The mesh is exported to FLUENT along with the physical properties and the initial conditions specified. The material properties and the initial conditions are read through the case file. Instructions for the solver are provided through a `journal file`. When the solution is converged or the specified number of iterations is met, FLUENT exports data to a `neutral` file and to `.xy` plot files. GAMBIT reads the `neutral` file for postprocessing.

3 Scope and Limitations

At higher angles of attack, a large separation region in the downstream of the airfoil is observed. This separation region makes the flow unsteady. Because the time dependent features of the flow field are neglected in this simulation, the flow predictions for higher angles of attack will not be accurate.

This exercise is aimed for resolving high-speed flows only. It is not possible to solve the flow system with laminar conditions. Difficulty in obtaining convergence or poor accuracy may result if input values are used outside the upper and lower limits suggested in the problem overview.

4 Exercise Results

4.1 Reports

The following reports are available:

- Wall shear stress *
- Skin friction factor *
- Coefficient of lift
- Coefficient of drag

* *Available only for viscous flow.*

4.2 XY Plots

The plots reported by FlowLab include:

- Residuals
- Pressure coefficient along airfoil
- Friction coefficient along the airfoil *
- Shear stress distribution along the airfoil *
- Wall Yplus distribution along the airfoil *

* *Available only for viscous flow.*

Figure 4.1 represents the plot of pressure coefficient along the airfoil.

4.3 Contour Plots

Contour plots of pressure, total pressure, temperature, velocity magnitude, Mach number, stream function, x-velocity, and y-velocity are available for both viscous and inviscid flows. Turbulent kinetic energy and dissipation rate are available for viscous flows. Contours of static pressure are presented in Figure 4.2.

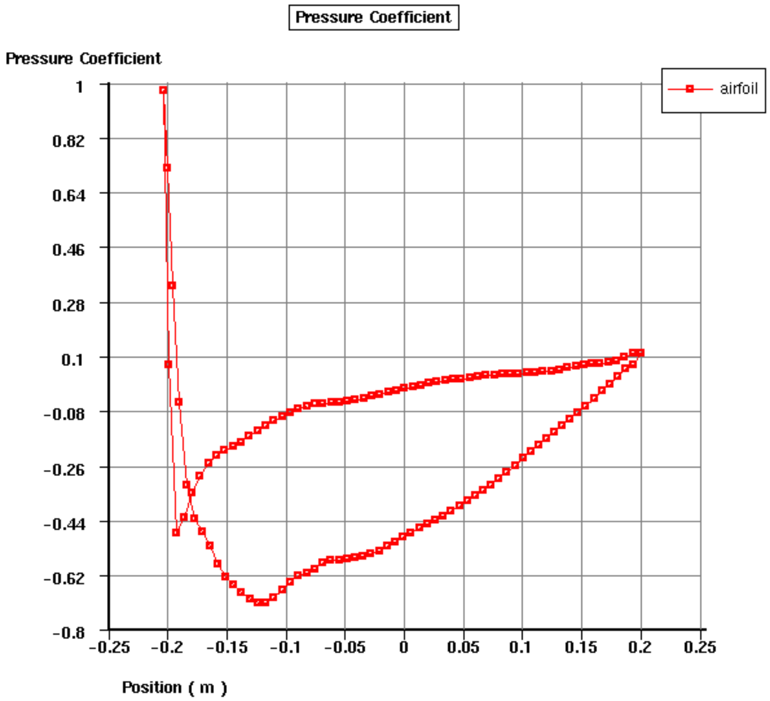

Figure 4.1: Plot of Pressure Coefficient along the Airfoil

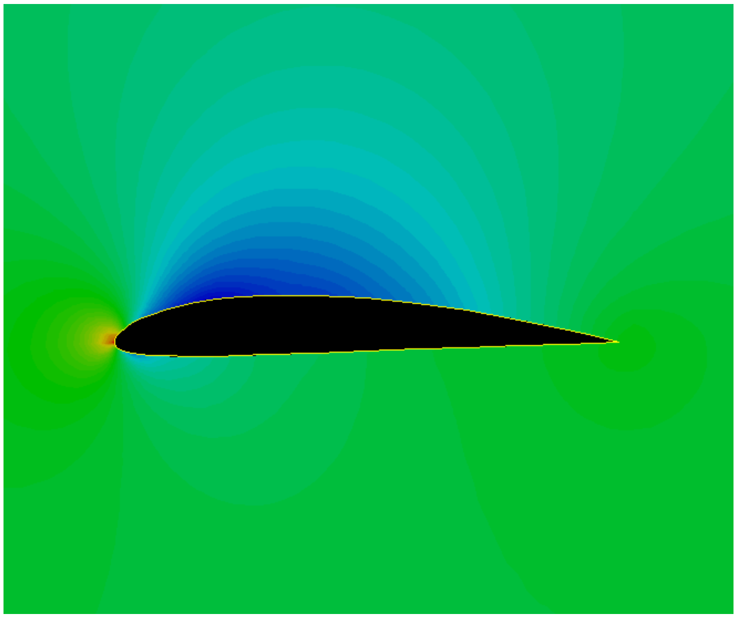

Figure 4.2: Contours of Static Pressure for the Default Case

5 Verification of Results

This exercise allows you to calculate lift coefficient at various angles of attack. Comparing lift coefficient with the angle of attack indicates a stalling angle of about 13 degree at a Reynolds number of 1.0e+5. These results were obtained for a Mach number of about 0.025 using the fine mesh option. For this same Reynolds number, the stalling angle for this airfoil as reported by experiment is approximately 12 degree [2].

Notes:

1. *Figure 5.1 was created using the XY plot routine within* FlowLab. *It is not a standard plot available in this exercise. A similar plot may be created using any spreadsheet by recording lift coefficient from the* Reports *form for different runs, with varying angle of attack.*
2. *Data provided by University of Iowa as per [2].*

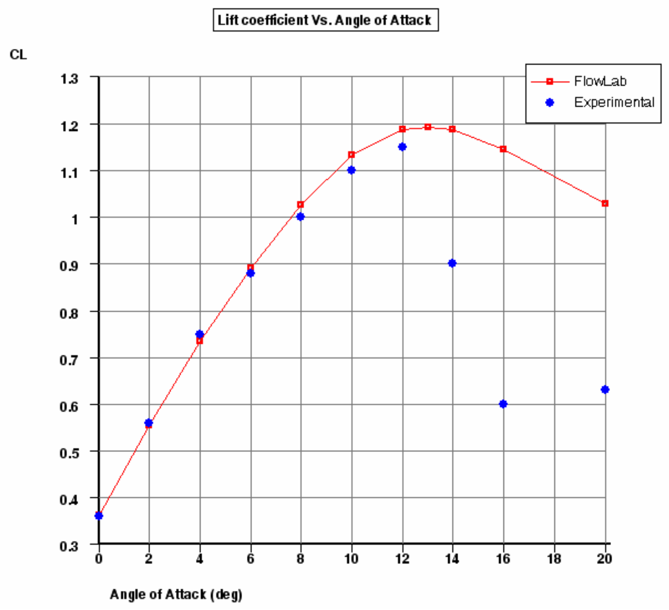

Figure 5.1: Variation of Lift Coefficient versus Angle of Attack, Re = 1.0e+5

6 Sample Problems

1. Run the default case, which simulates a free stream Mach number of 0.03 at 0^0 angle of attack.
2. For the same far-field pressure, temperature, and Mach number, vary the angle of attack in the range of -5.0^0 to $+20^0$. Observe the effect of angle of attack on c_l and c_d and plot these coefficients as a function of angle of attack (Figure 5.1).
3. Repeat the same procedure as described in Problem 2 for a range of Mach numbers.
4. Investigate the effect of changing far field pressure and temperature values while observing their impact on c_l and c_d.

7 Reference

[1] *John. D Anderson Jr.*, "Fundamentals of Aerodynamics", McGraw Hill, 1991

[2] *Marcman J. F. Virginia Polytechnic Institute*, "Clark-Y airfoil performance at low Reynolds numbers", AIAA-84-0052