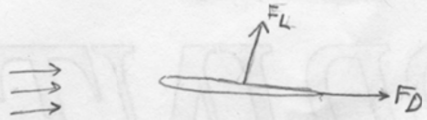
MAE 4230/5230 Homework 9 Solutions

(1.) Consider incompressible, inviscid flow over an airfoil at low angle of attack



The net force exerted by the fluid on the airfoil is the sum of two components, the viscous force and the pressure force. We may write the lift force (F_L) and the drag force (F_d) in terms of these 2 components:

$$F_L = (F_L)_{pressure} + (F_L)_{visc} \quad ; \quad F_D = (F_D)_{pressure} + (F_D)_{visc}$$

where $(F_L)_{pressure}$ e.g. is the component of pressure force in the lift direction. Because the flow is inviscid, there is no viscous or shear contribution to either the lift / drag force. This has only a minimal impact on the lift force but a large effect on the drag force, that is there is only small component of $(F_L)_{visc}$ and almost all the viscous force is in the drag direction. Therefore we can write:

$$F_L = (F_L)_{pressure} \quad ; \quad F_D \neq (F_D)_{pressure}.$$

While no boundary layer exists in the simulation, the effects of neglecting it, w.r.t. the lift are acceptable since there is little pressure variation across the boundary layer. ∴ The lift force in FLUENT should agree well w/ experiments. However, at low angles of attack, there is almost no contribution of the pressure force in the drag direction, so that $(F_D)_{pressure} \approx 0$.

For more detailed look, see an article on: D'Alembert's paradox

(2.) $$\rho\left[\frac{\partial \vec{u}}{\partial t} + \vec{u}\cdot\nabla\vec{u}\right] = -\nabla p + \mu \nabla^2 \vec{u}$$

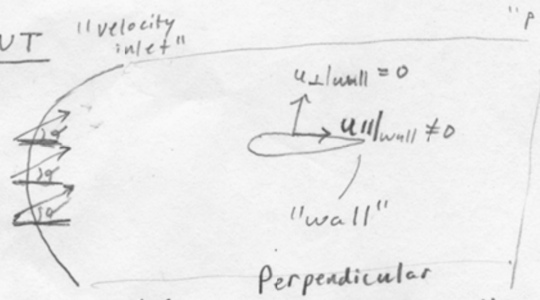
steady; inviscid

$\rightarrow \quad \rho\vec{u}\cdot\nabla\vec{u} = -\nabla p$ (steady Euler Equation)

continuity (incompressible)

$$\nabla\cdot\vec{u} = 0$$

FLUENT



Perpendicular component of velocity at wall is 0.

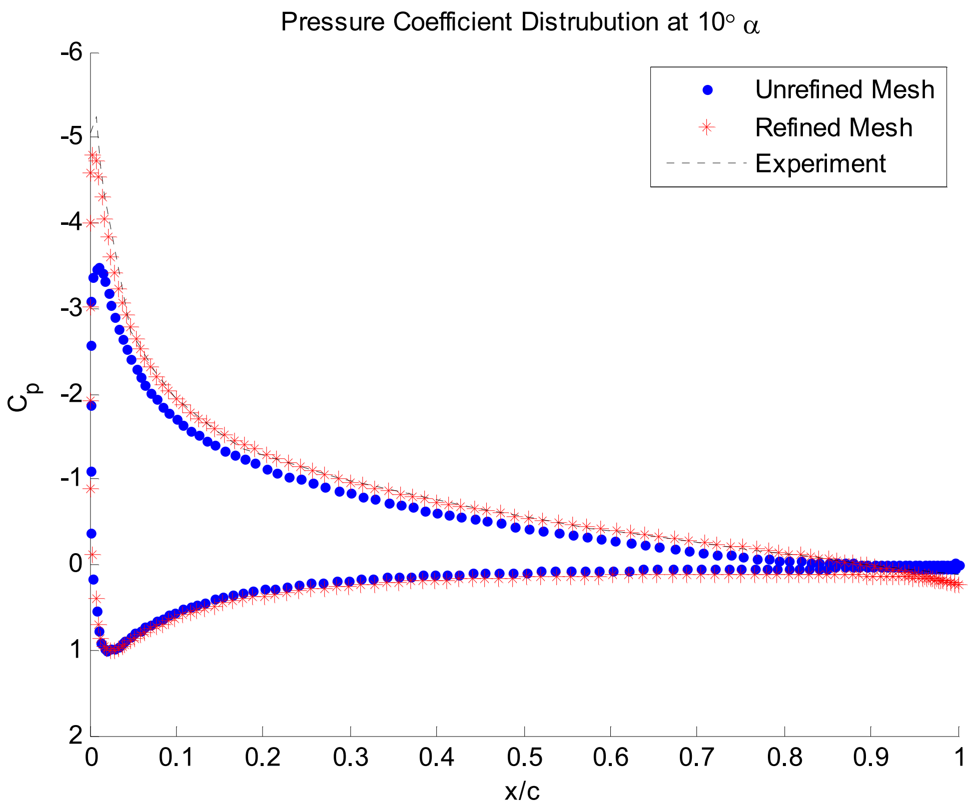
*) In FLUENT, we retain the airfoil at a horizontal position and incline the inlet flow.

Velocity components:

$$\left.\begin{aligned} u_x &= 1\,\text{m/s}\cos 10^\circ \\ u_y &= 1\,\text{m/s}\sin 10^\circ \end{aligned}\right\}$$

3.

(a)



(b)

Mesh/α	C_L (FLUENT)	C_D (FLUENT)
10 degrees unrefined	.8713	.0281
10 degrees refined	1.0822	.01584
6 degrees unrefined	.6430	.01225

α	C_L (Experiment)	C_D (Experiment)
10 degrees	1.089*	.0138
6 degrees	.663	.0090

*Typo in HW handout; this value referenced from Caughey pdf file.

$$\%\ difference = \frac{|FLUENT\ Result - Experimental\ Result|}{Experimental\ Result} \times 100$$

10 degrees, unrefined mesh:

$$\%\ \Delta C_L = \frac{|.8713 - 1.089|}{1.089} \times 100 = 19.99\%$$

$$\%\ \Delta C_D = \frac{|.0281 - .0138|}{.0138} \times 100 = 103.6\%$$

10 degrees, refined mesh:

$$\%\ \Delta C_L = \frac{|1.0822 - 1.089|}{1.089} \times 100 = .624\%$$

$$\%\ \Delta C_D = \frac{|.01584 - .0138|}{.0138} \times 100 = 14.78\%$$

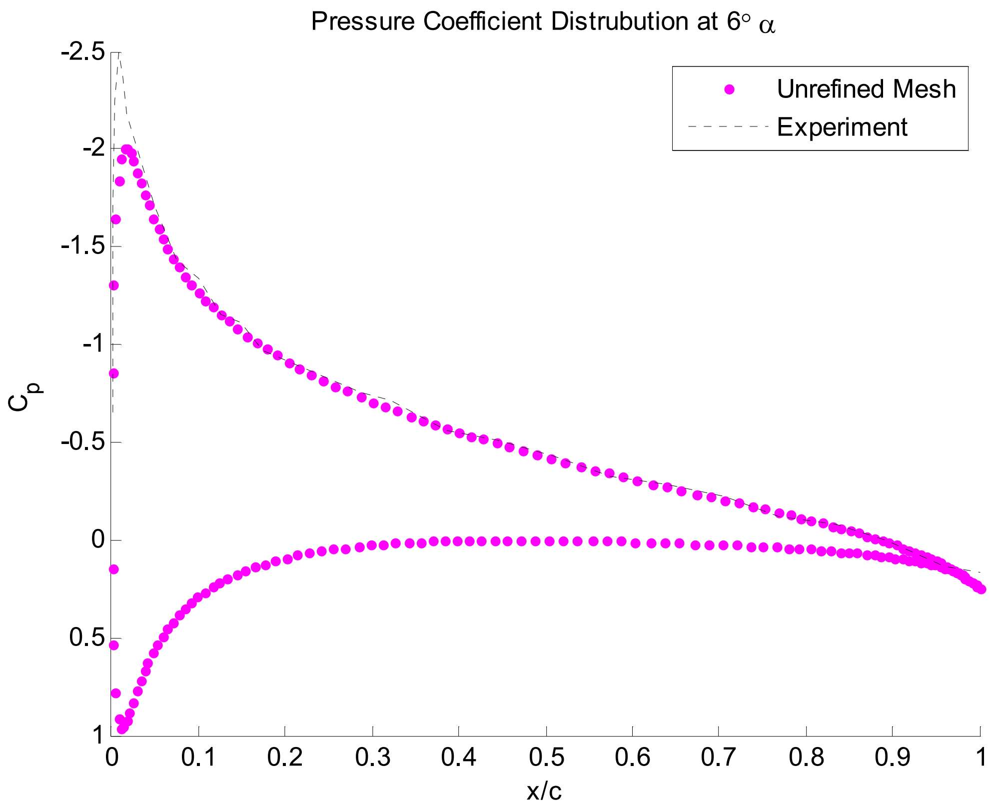
6 degrees, unrefined mesh:

$$\%\ \Delta C_L = \frac{|.6430 - .663|}{.663} \times 100 = 4.55\%$$

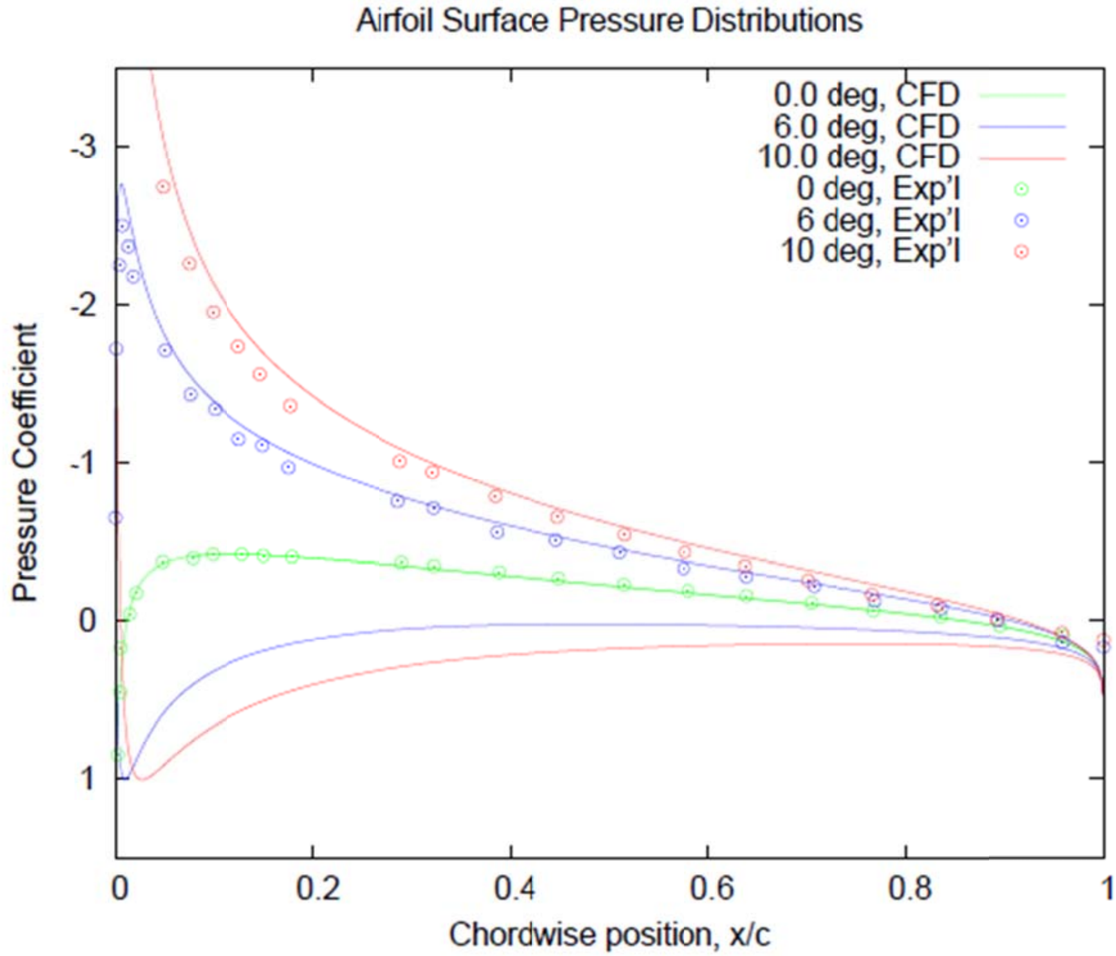
$$\%\ \Delta C_D = \frac{|.01225 - .0090|}{.0090} \times 100 = 36.11\%$$

One mesh refinement for the 10 degree case clearly yielded lift and drag coefficients that agreed better with the experimental data than when experimental results were compared with data using the unrefined mesh. Drag coefficients overall did not line up well with experimental results. Because of the inviscid assumption used in FLUENT, the drag coefficients obtained from the simulation neglect the component of drag due to viscosity which makes up a larger component of the total drag than does the pressure drag, at least for low angles of attack (which we studied in these two cases). Note however that FLUENT does yield drag coefficients that are the correct order of magnitude even with the inviscid assumption.

(c)


Pressure Coefficient Distrubution at 6° α
Unrefined Mesh
Experiment
C_p
-2.5
-2
-1.5
-1
-0.5
0
0.5
1
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
0.9
1
x/c

(d)
(Prof. Caughey's results)



We notice the pressure coefficient distribution looks qualitatively similar between the 6 and 10 degree angle of attack cases, with the magnitude of Cp for the 10 degree case being somewhat larger than the 6 degree case. The FLUENT simulation results look qualitatively similar to Prof. Caughey's results. We notice that the pressure coefficient curve becomes steep at the trailing edge of the airfoil. Potential theory predicts an infinite pressure gradient at the trailing edge of an airfoil at non-zero angle of attack as the fluid leaving the trailing edge must instantaneously change direction to match the free stream vectors. Prof. Caughey's mesh was likely more fine than either of the meshes used in FLUENT, thereby allowing Prof. Caughey's code to construct a more "realistic" potential flow solution.

MATLAB code:

```
clear all
close all
clc

Cp10_unref = dlmread('Cp_10deg_unref','\t',[4 0 203 1]);
Cp10_ref   = dlmread('Cp_10deg_ref','\t',[4 0 203 1]);
Cp6_unref  = dlmread('Cp_6deg_unref','\t',[4 0 203 1]);
Cp6_exp    = dlmread('cp_06degExperiment1.txt');
Cp10_exp   = dlmread('cp_10degExperiment1.txt');

hold on
plot(Cp10_unref(:,1),Cp10_unref(:,2),'b.')
plot(Cp10_ref(:,1),Cp10_ref(:,2),'r*')
plot(Cp10_exp(:,1),Cp10_exp(:,2),'k:')
set(gca,'YDir','reverse')
title('Pressure Coefficient Distrubution at 10\circ \alpha')
xlabel('x/c')
ylabel('C_p')
legend('Unrefined Mesh','Refined Mesh','Experiment')
figure
hold off
hold on
plot(Cp6_unref(:,1),Cp6_unref(:,2),'m.')
plot(Cp6_exp(:,1),Cp6_exp(:,2),'k:')
set(gca,'YDir','reverse')
title('Pressure Coefficient Distrubution at 6\circ \alpha')
xlabel('x/c')
ylabel('C_p')
legend('Unrefined Mesh','Experiment')
```