EAE 127 Applied Aircraft Aerodynamics

Project 5
Viscous Effects

Upload all files to the 'Assignments' section on Canvas in a single, compressed (zip) folder, which must contain a '.ipynb' Jupyter Notebook report (all Python code must run), a '.html' hard copy, and all data files necessary to run code. 'Run All' before uploading. (More details: 'EAE127_FAQ.pdf'). DUE: Monday 11/23/20 11:59pm

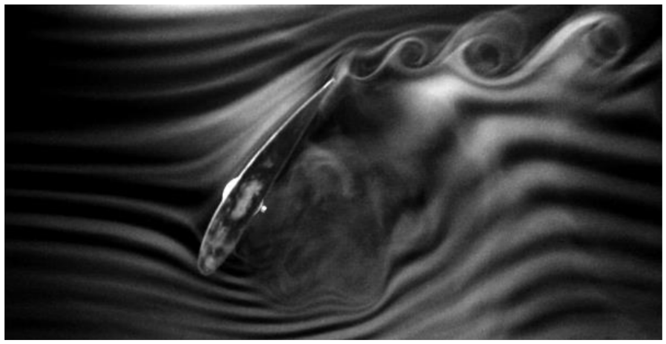

Fig. 1: Streamlines in separated flow over an airfoil

1 Boundary Layer Properties

Let's start our investigation into viscous flow by taking a look at viscosity's most famous flow effect: the boundary layer. We will use the results of Blasius' solution for incompressible flow over a flat plate (Anderson 18.2) as a simplified model of the T-38's UHF antenna (see Fig. 2).

Fig. 2: A NASA T-38 aircraft variant with a large UHF antenna mounted to the nose (red arrow)

1.1 Flight Conditions

Consider two flight conditions of the T-38: taxing on the tarmac and cruise.

Table 1: **T-38 Flight Conditions**

	BL Type	Altitude (ft)	Velocity (kts)
Taxi	Laminar	0	12
Cruise	Turbulent	32000	575

Assume a constant viscosity μ equal to the value at sea level and that we can ignore the effects of compressibility. The width of the antenna (streamwise dimension) is $6in$. **Compute and report the Reynolds number** of the antenna for each case; **make sure each is appropriate** for the boundary layer type specified above. **How valid is our assumption of incompressible flow** for each case?

Note: Boundary layer transition is not an exact science, but we can use a rule-of-thumb transition Reynolds number (critical Re) to help use judge if a flow is laminar or turbulent. For this problem, use $Re_{cr} = 5e5$ to determine boundary layer transition (See Anderson Section 15.2).

1.2 Boundary Layer Velocity Distribution

Next, we will take a look at the distribution of velocity within the boundary layer on the antenna at multiple flight conditions. Empirical boundary layer analysis has provided us with approximations of the vertical profile of the streamwise (horizontal) velocity component within both laminar and turbulent boundary layers:

$$\left.\frac{u}{u_e}\right|_{lam} \approx \left(\frac{2y}{\delta} - \frac{y^2}{\delta^2}\right); \quad \left.\frac{u}{u_e}\right|_{turb} \approx \left(\frac{y}{\delta}\right)^{\frac{1}{7}} \tag{1}$$

where u is the streamwise velocity at some point above the surface and below the edge of the boundary layer and u_e is the streamwise velocity at the edge of the boundary layer.

Plot both boundary layer velocity profiles in two figures (like Fig. 3):

- **Dimensional velocity** u **as vs. dimensional height** y at the aftmost edge of the boundary layer
- **Non-dimensional velocity** $\frac{u}{u_e}$ **vs. non-dimensional height** $\frac{y}{\delta}$

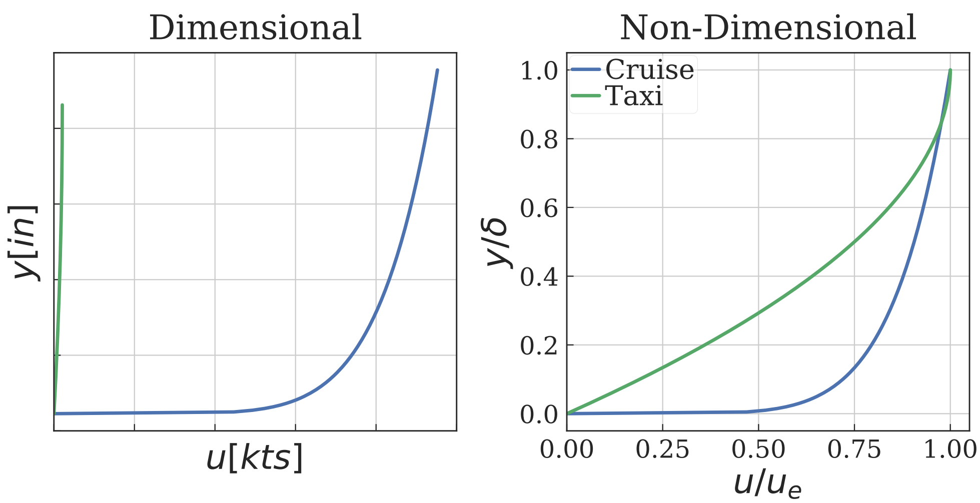

Fig. 3: Boundary layer vertical distributions of streamwise velocity (your plot will have numbers on the ticks)

To compute dimensional values, assume the boundary layer edge velocity u_e is the same as the surrounding freestream and use the laminar and turbulent Blasius solution approximations for boundary layer thickness of a flat plate:

$$\delta(x)_{lam} = \frac{5.0x}{\sqrt{Re_x}}; \quad \delta(x)_{turb} = \frac{0.16x}{Re_x^{\frac{1}{7}}} \tag{2}$$

Discuss the differences between the two boundary layer velocity profiles. Can you explain (using your plot) **why a turbulent boundary layer is more resistant** to the effects of an adverse pressure gradient? Why does the **turbulent boundary layer impart a greater skin friction force**?

1.3 Friction Drag

Using the flat plate boundary layer approximation, **compute the total drag coefficient due to friction** C_f on the entire antenna for both flight conditions. Use Anderson Eqns 18.22 and 19.2:

$$C_{f,lam} = \frac{1.328}{\sqrt{Re_c}}; \quad C_{f,turb} = \frac{0.074}{Re_c^{\frac{1}{5}}} \tag{3}$$

(Remember: a flat plate has one side; an antenna fin has two...). What are the **friction drag forces on the aircraft due to the fin** if the fin is $12in$ long (extending from the fuselage)? **Is this significant**?

2 Viscous/Inviscid Interaction

Next, we will apply our analytic boundary layer models to our potential flow panel method solutions to create a first attempt at an approximation of true viscous flow.

This will be done in an iterative process, where the displacement thickness δ^* will be computed from the surface velocity distribution of a potential flow solution (panel method) and then the original geometry will be modified to account for the streamline displacement caused by the viscous boundary layer. The potential flow solution can then be re-computed on the new geometry, which should be more representative of the actual flow (outside the boundary layer). A much more sophisticated implementation of this process is used in XFOIL to provide viscous results.

You have been provided with an example code that runs an inviscid XFOIL simulation of flow over an airfoil and then uses the resulting pressure distribution to calculate the boundary layer momentum thickness (Eqn 4). This relation is derived by applying the Blasius solution to the von Karman integral relation (see provided supplemental notes).

$$[\theta(x_0)]^2 = \frac{0.440\mu}{\rho[u_e(x_0)]^{9.210}} \int_0^{x_0} [u_e(x)]^{8.210} dx \tag{4}$$

2.1 Dividing Streamline Displacement

You must use this code to create a Viscous/Inviscid Interaction (VIvI) tool. (Note: XFOIL and pyxfoil can be run in the CAE if you cannot get them to work on your personal computer). Wrap the provided code in a loop so that you can feed its "viscous" solution back into XFOIL and repeat the process as many times as specified. **Describe the pseudo-code of Viscous/Inviscid Interaction** to demonstrate your understanding of the process.

First, for each potential flow solution, you will need to calculate displacement thickness δ^* from momentum thickness θ using the Blasius result for the shape factor H:

$$H = \frac{\delta^*}{\theta} = 2.605 \tag{5}$$

For a NACA 23012 airfoil at zero angle of attack $\alpha = 0^o$, **plot** the **original airfoil geometry and the "displaced" geometry for two** more iterations in the same figure, in a manner similar to Fig 4.

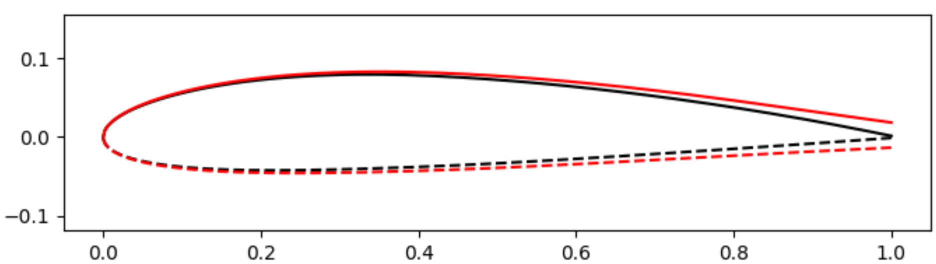

Fig. 4: An airfoil (black) and the dividing streamline displaced by its boundary layer (red)

What trends do you observe with the geometry as a function of iteration? **Does the geometry converge** with more iterations or does it diverge and require a pre-chosen iteration limit?

Demonstrate and discuss the concept of viscous de-cambering by computing the camber lines for each ViVI iteration (average the upper and lower surfaces) and **co-plotting all camber lines in two separate figures:**

1. **Airfoil surface geometries *and* camber lines** of each iteration, same color (Use *plt.axis('equal')*). You may combine this and the "displaced" geometry plot if it is properly labeled and not too busy
2. **Only camber lines** for each ViVI iteration (Don't use *plt.axis('equal')* to emphasize the differences)

2.2 Viscous Force Coefficients

For the 3rd iteration (geometry after running XFOIL and adding the displacement thickness 3 times), **compute the lift and drag coefficients** C_l, C_d.

Run an inviscid XFOIL simulation with the 3rd iteration geometry to determine C_l. Note: You will receive no points for 0th iteration results (direct output of the provided code). You must create the logic to loop through the code multiple times.

For C_d, the XFOIL contribution from pressure drag will be zero since this is potential flow. Instead, we will compute the viscous contribution from friction ($C_d = C_f$), which can be accomplished by integrating the wall shear stress τ:

$$C_f = \frac{1}{c}\int_0^c c_f dx; \quad c_f = \frac{\tau}{\frac{1}{2}\rho_\infty V_\infty^2} \tag{6}$$

(see Anderson Eqns 18.19, 18.21).

We will need to **derive an expression** for wall shear stress τ in terms of momentum thickness θ from the Blasius results (Hint: relate the following equations via Reynolds number):

$$\tau = 0.332\frac{\mu u_e}{x}\sqrt{Re_x}; \quad \theta = \frac{0.664x}{\sqrt{Re_x}} \tag{7}$$

Compute τ at every x location along the airfoil using the θ that results from the pressure distribution obtained in the XFOIL simulation used to calculate C_l, then integrate according to Eqn 6 to obtain drag coefficient.

In a table, **compare your results** for C_l, C_d to **inviscid XFOIL results** with the original airfoil geometry as well as **viscous XFOIL results** at the Reynolds number corresponding to this simulation ($c = 1m, V_\infty = 1m/s, \rho_\infty = 1.225kg/m^3, \mu = 1.79e-5Pa\cdot s$). **Report your result for Reynolds number**.

How does each coefficient change between the inviscid XFOIL result and your VIvI solution? **Explain the reasoning** behind each. **How close does our VIvI simulation come to viscous XFOIL**? (Remember, XFOIL uses a much higher fidelity model than this method).

2.3 Assumptions and Limitations

Anything called a "model" is only an approximation of reality. When we use an approximate model like Viscous/Inviscid Interaction, it is important to be aware of the conditions under which its results should be considered unrealistic.

What are the limitations of this VIvI tool? (Remember: Flow results come from an inviscid panel method solver and viscous results come from the **Blasius solution** for a flat plate). **What flow conditions would this method be inappropriate for**? **What different methods could be used for higher-fidelity modeling of viscous effects** in these situations?