Cover Page for Precalculations – Individual Portion

Introduction to Computational Fluid Dynamics

Prepared by Professor J. M. Cimbala, Penn State University
Latest revision: 17 January 2017

Name: _______________________________________

Date: _______________________________________

Section number: ME 325.______ Group # _____

Score (For instructor or TA use only):

| Precalculations | ______ / 30 |

Comments (For instructor or TA use only):
Precalculations

A. Flow over a Cylinder

In the first part of this lab, the instructional CFD code FLUENT (now part of ANSYS) will be used to calculate the drag force $F_D$ on a two-dimensional cylinder whose radius is 2 cm.

(4) 1. Calculate the drag coefficient $C_D$, per unit depth, given the drag force is 7.2 µN and a velocity of .0003 m/s. Note the flow is perpendicular to the axis of the cylinder (see the figure on page 8 of the Tutorial appendix available on the course website).

(3) 2. In the first CFD exercise, where should the computational mesh be the finest and why?

(3) 3. When choosing a mesh size there are tradeoffs one must consider. What are the tradeoffs of picking a finer mesh over a courser one?

B. Laminar Boundary Layer Flow over a Flat Plate

In the second part of this lab, you will use the CFD code FLUENT (now part of ANSYS) to simulate the growth of a boundary layer along a flat plate aligned parallel to the flow. Laminar flow is assumed. You will generate the computational domain, specify boundary conditions, and generate the grid using the software provided in ANSYS. The physical geometry and the computational domain are sketched in Figure 8.

Figure 8. (a) physical geometry and (b) computational domain (blue shaded area) for CFD calculation of the boundary layer growing on a flat plate (not to scale).
1. For each numbered face on the computational domain of Figure 8b, specify the most appropriate boundary condition:

   Boundary condition 1 = ____________________________.

   Boundary condition 2 = ____________________________.

   Boundary condition 3 = ____________________________.

   Boundary condition 4 = ____________________________.

   Boundary condition 5 = ____________________________.

2. The CFD calculations for flow over a laminar flat plate boundary layer will be compared to the simple similarity solution of Blasius (1908). His solution is in terms of nondimensional variables \( \frac{u}{U_\infty} \) as a function of \( y \left( \frac{U_\infty}{v} \right)^{1/2} \). Because of this clever nondimensionalization, any laminar flat plate boundary layer for any Newtonian fluid at any Reynolds number should collapse onto the same universal Blasius profile. Fill in the blank spaces to complete this predicted laminar flat plate boundary layer velocity profile, and show where you obtained your data:

   **Table 1.** Blasius velocity profile in nondimensional form (valid for all flat plate laminar boundary layers).

   \[
   \begin{array}{ccc}
   \sqrt{y \left( \frac{U_\infty}{v} \right)} & \frac{u}{U_\infty} \\
   0.0 \quad \text{(at the wall)} & 0.0 \quad \text{(no slip condition)} \\
   0.2 & 0.06641 \\
   0.4 & 0.13277 \\
   0.6 & 0.19894 \\
   0.8 & 0.32979 \\
   1.0 & 0.39378 \\
   1.2 & 0.51676 \\
   1.4 & \_\_\_\_\_ \\
   1.6 & 0.84605 \\
   1.8 & 0.87609 \\
   2.0 & 0.90177 \\
   2.2 & 0.94112 \\
   2.4 & 0.95552 \\
   2.6 & 0.96696 \\
   2.8 & 0.97587 \\
   3.0 & 0.98269 \\
   3.2 & 0.98779 \\
   3.4 & 0.99155 \\
   3.6 & \_\_\_\_\_ \\
   3.8 & \_\_\_\_\_ \\
   4.0 & \_\_\_\_\_ \\
   4.2 & \_\_\_\_\_ \\
   4.4 & \_\_\_\_\_ \\
   4.6 & \_\_\_\_\_ \\
   4.8 & \_\_\_\_\_ \\
   5.0 & \_\_\_\_\_ \\
   \infty \quad \text{(far outside the boundary layer)} & 1.00000 \quad \text{(freestream velocity)}
   \end{array}
   \]

   Data are from the following Reference (author, title, publisher, copyright date, page number(s), table number, etc.):
(5) 3. Create a plot (Excel is recommended) of the Blasius boundary layer profile. The horizontal axis should be the normalized axial velocity \( u/U_\infty \), and the vertical axis should be the similarity variable \( \left( \frac{U_\infty}{V_x} \right)^{1/2} \). Use a line (no symbols) for your plot, and attach it to your Precalculations. After the lab is finished, you will plot the data predicted by CFD on this same plot (as symbols, no line), so be sure to save your Excel file for future use.

See Figure __________.

(2) 4. For the computer code, the grid over a two-dimensional flat plate will be generated. The freestream velocity \( U_\infty \), above the flat plate is specified as 0.10 m/s. The plate length is \( L = 0.50 \) m in the streamwise direction. Assuming the fluid is water at 20°C (density is nearly 1000 kg/m\(^3\) and kinematic viscosity is about \( 1.0 \times 10^{-6} \) m\(^2\)/s), estimate the Reynolds number \( Re_x \) at the end of the plate, showing your calculations below:

\[ Re_x = \] _________________

(3) 5. For the conditions above, and assuming the boundary layer remains laminar, estimate the laminar boundary layer thickness at the end of the plate, showing your calculations below:

\[ \delta = \] _________________ mm.

**Experimental Objectives**

- Get familiar with operation of the CFD code FLUENT in ANSYS Workbench.
- Determine the velocity and pressure fields produced when a fluid flows over a cylinder and calculate drag force exerted by the fluid over the cylinder using the CFD code FLUENT.
- Generate a grid for calculation of a laminar flat plate boundary layer, using the grid generating code in ANSYS.
- Define the boundary conditions, fluid properties, and numerical properties for flow over a flat plate using the CFD code FLUENT.
- Calculate laminar boundary layer profiles on a flat plate using FLUENT, and compare to published data, namely those of Blasius.