MASSACHUSETTS INSTITUTE OF TECHNOLOGY
DEPARTMENT OF MECHANICAL ENGINEERING
CAMBRIDGE, MASSACHUSETTS 02139

2.29 NUMERICAL FLUID MECHANICS— SPRING 2007

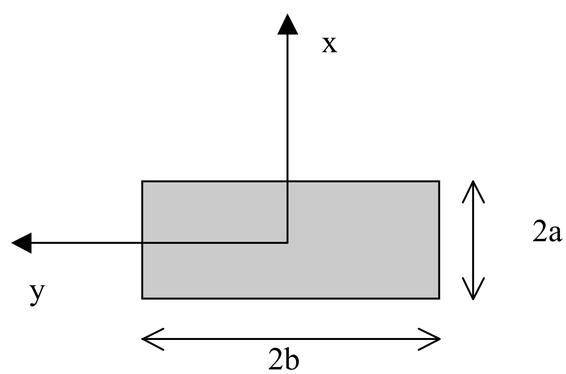
Problem Set 4

Totally 120 points
Posted 04/21/07, due Thursday 4 p.m. 05/08/07, Focused on Lecture 18 to 22

Problem 4.1 (120 points): Steady state unidirectional fully developed incompressible laminar flow in a rectangular pipe

a) Write down the Navier Stokes equation for the fully developed incompressible laminar flow in a pipe with arbitrary constant cross section and simplify the equations[1]. Furthermore simplify the equation for the case when the pressure gradient is fixed. Explain when your assumption holds.

b) What is the name of this equation? Categorize that and give at least five examples of other cases (possibly from other physical domains) where we encounter the same equation.



Now consider the upper rectangular pipe where $a \le b$.

c) Find an analytical solution by separation of variables.

d) (EXTRA CREDIT 10 Points) Explain how your solution (method) changes if we had a pulsating flow. You do not need to solve it thoroughly.

[1] If you have difficulty in deriving the equations you can look at different basic textbooks. In particular "Analysis of Transport Phenomena" by W. M. Deen can be suggested.

e) Compute the maximum shear stress and the net volumetric flow rate.

f) Compare the maximum shear stress and the pressure gradient in a square pipe with a circular pipe with the same area and the same flow rate.

By now we have solved the equation analytically for our simple cross section, but in general it will be very hard or almost impossible to solve it analytically for arbitrary cross sections. Indeed we usually have to rely on numerical methods and here we will develop a finite difference method to solve the equation.

g) Develop a finite difference scheme to solve the equation for the square pipe. Start by a mesh consisting of 2 elements (3 nodes) in each directions and refine it in each step by a factor of two until your net flow rate computed has a relative error of less than 0.1% compared to the previous step. Use the proper integration for computing the flow rate.

h) Compare the flow rate computed above with the analytical solution.

i) Plot the flow rate as a function of mesh size and discuss the curve slope.

j) Plot the numerical and analytical velocity contours.

k) Due to Laplacian operator in the equations it sounds appealing to use a uniform grid. However, it is not always possible to use a uniform grid, especially in extreme cases like when $\frac{a}{b} \to 0$. To manipulate those cases it is very good to nondimensionalize the equation. So nondimensionalize the equation with $\eta = \frac{x}{a}, \xi = \frac{y}{b}$.

l) Solve the finite difference equation of previous part in (η, ξ) domain for the case where "b=10a". Use the same number of elements as the last step of part g.

m) Compute the flow rate of previous part and compare it to the analytical solution. Also plot the velocity contours as well as shear stress contours.

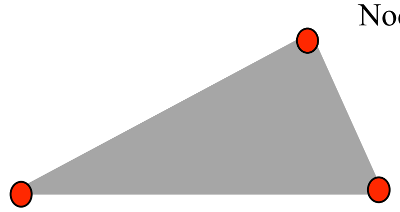
n) Use the MATLAB PDE tool and solve the equation of part k with a mesh preferably as much refined as part "l". Explain clearly how you compute the volumetric flow rate and repeat part "m".

o) (EXTRA CREDIT 10 Points) Now consider the case where $\frac{a}{b} \to 0$. Compute the volumetric flow rate normalized by "b" (flow rate per unit of depth) from analytical solution. Compare it with similar 1D problem and discuss whether they are equivalent or not. Also discuss what happens in the numerical solution and how it changes.

Problem 4.2 (0 points): Elementary Ingredients of Panel Methods

You do not need to turn in this homework but make sure that you have the required setup for the below list so you can use it for “quiz 2”. Also note that the solution of this part will be uploaded after “quiz 2” but here you will build some ingredients of panel methods.

Assume that we have a triangulated 3D surface, possibly making a closed object. This means that we have a set of numbered nodes in 3D Cartesian coordinate and a set of numbered elements each referring to three ordered nodes. Nodes of each elements are ordered such that the they all point to the outer surface[2].



a) Plot the 3D surface.

b) Compute the area of each element.

c) Be able to integrate a scalar field on the area ($\int F(x, y, z)dA$).

d) Plot a scalar value on the 3D surface.

e) Compute the normal vector of each element.

f) Be able to integrate a vector field on the area ($\int \vec{F}(x, y, z).\vec{n}_A dA \;\; or \;\; \int \vec{F}(x, y, z)dA$).

[2] More precisely this means that the vector $\vec{N} = (\vec{R}_2 - \vec{R}_1) \times (\vec{R}_3 - \vec{R}_2)$ points outward of the surface ($\vec{R}_i$ refers to the coordinate of i[th] node of the element).