MME9710a Assignment # 4

Date Given: November 10, 2015 Date Due: November 24, 2015

This assignment will give you experience developing and coding an algorithm to solve the coupled mass and momentum equations for 1D fluid flow problems. For this assignment, we won't be concerned with heat transfer, so the energy equation does not need to be coupled with mass and momentum. The subroutines that you are required to write (and implement) are given below.

File	Description	Status
gradp.f:	computes derivatives of `P` field	outline provided
srcu.f:	computes source terms in `U` equation	outline provided
coeffm.f:	assembles active coefficients for `U` equation	outline provided
srcup.f:	adds the pressure source term to the `U` equation	outline provided
bndcu.f:	applies boundary conditions in the `U` equation	outline provided
dhat.f:	computes coupling coefficients	outline provided
adcont.f:	inserts the mass coefficients into the `P U` blocks	outline provided
bndcp.f:	applies boundary conditions on `P`	outline provided
uhat.f:	computes `UHE`	outline provided
nullvb.f:	nulls a vector block	complete
solup.f:	organizes solution of `P U` set	complete
globms.f:	creates the global coefficient matrix	complete
lud.f:	direct solver for global coefficient matrix	complete
assign.f:	transfers the direct solution to `U P` arrays	complete
residm.f:	computes residuals from block matrix equation	complete

Debugging Tips:

- set `DI=RHO=MU=U=UHE=DTIME=1` in the *in.dat* file; modify *makgrd.f* to give `AREP=1`; this way all of the coefficients have very simple values and problems in calculations tend to jump out at you.
- write out everything while you are debugging and check the computed values with hand calculations!

Problems

Solve all problems for water with $\rho = 1000$ $[kg/m^3]$, $\mu = 1 \times 10^{-3}$ $[kg/m \cdot s]$. Problems 1–3 should be solved for a 4 $[m]$ long, 0.02 $\times$ 0.02 $[m]$ cross–section duct discretized using 10 equal–length control–volumes. Also, for simplicity, use UDS as your advection scheme for problems 1–2. Higher–order advection schemes are considered in question 3.

1. For $\tau_w = 0$, set `U=UHE=10` $[m/s]$ and `P=0` $[Pa]$ everywhere as initial conditions. These are the exact solutions to the mass and momentum equations for constant duct area. Do one time step (using any time step size) and make sure that your code accepts

this as the exact solution. Repeat this problem with `U=UHE=-10` $[m/s]$ to ensure that your code has no directional dependence. Describe the boundary conditions used for the two cases.

2. For turbulent flow in a long duct, the wall shear stress can be approximated by:

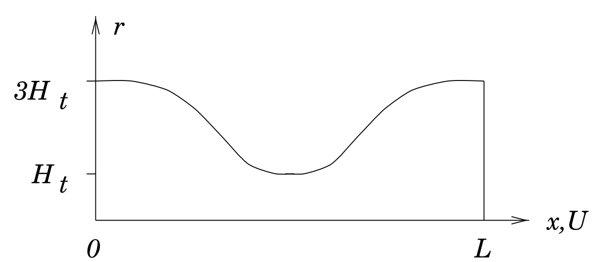
$$\frac{\tau_w}{\frac{1}{2}\rho U^2} = C_f = (1.58\ln(Re) - 3.28)^{-2}$$

where $Re = \rho D_h U/\mu > 10^4$ and $D_h = 4A/P_o$ is the hydraulic diameter of the duct. Implement the wall shear stress model into *srcu.f* using Newton–Raphson linearization. Note that the force on a control–volume can be computed using the above expression as:

$$\vec{F}_u = \tau_w A_o = C_f \frac{1}{2}\rho U^2 A_o$$

Note also that C_f changes very slowly with U, so use Newton–Raphson linearization only on the U^2 term. Impose suitable boundary conditions on the ends of the duct and initialize the problem with `U=UHE=10` $[m/s]$ and `P=0`. Check that the pressure is exactly correct after emerging from enough `P-U` iterations, i.e. compare with the exact solution calculated from the above expressions. Why isn't the result correct after the first `P-U` iteration?

3. In this problem, we explore the flow in a frictionless converging–diverging circular duct. We will now consider the utility of second–order advection schemes to explore errors associated with UDS. Implement the CDS and QUICK schemes into the momentum equation. You can essentially use what you created in the previous assignment for this task. The duct is defined by:



$$r = 2H_t + H_t cos\left(2\pi\frac{x}{L}\right)$$

where $L = 1$ $[m]$, $H_t = 0.01$ $[m]$. Solve the problem for water with $\rho = 1000$ $[kg/m^3]$, $\mu = 1 \times 10^{-3}$. The inlet velocity should be imposed as `U=2` $[m/s]$. To eliminate friction in the duct, set the friction factor C_f to zero in the momentum equation (i.e. in *srcu.f*). Solve the problem using 8, 16, 32 and 64 equal–length control–volumes and calculate the loss in dynamic head from each converged solution. The dynamic head loss is given as:

$$C_D = \frac{P_{in} - P_{out}}{\frac{1}{2}\rho U_{in}^2}$$

Compare your solutions from UDS with those from the second–order schemes and quantify the convergence characteristics of each. Plot the velocities and pressures verses x for a few of the cases. What should C_D become for this problem?