ASEN 5327 Computational Fluid Dynamics
Spring 2009

Course Project, due Thursday, April 30

Reading: Sections 5.1, 5.2, 5.5, 6.1, 6.3 6.7 of Tannehill *et al.*

The two-dimensional Euler equations can be written as

$$\frac{\partial}{\partial t}\int_V U\, dV \;+\; \int_S \left[F\hat{e}_x + G\hat{e}_y\right]\cdot\vec{n}\, dS \;=\; 0 \tag{1}$$

where

$$U = \left\{ \begin{array}{c} \rho \\ \rho u \\ \rho v \\ E_t \end{array} \right\} \qquad F = \left\{ \begin{array}{c} \rho u \\ \rho u^2 + p \\ \rho u v \\ (E_t + p)u \end{array} \right\} \qquad G = \left\{ \begin{array}{c} \rho v \\ \rho u v \\ \rho v^2 + p \\ (E_t + p)v \end{array} \right\} \tag{2}$$

Modify your quasi-1D program to solve the two-dimensional Euler equations for the flow through a 2-D supersonic inlet, whose duct walls vary according to

$$y_w(x) = \pm y_t \left\{ 1 - \frac{1}{2}\sin^4\left[\pi\left(\frac{x}{L}\right)\right]\right\}; \qquad \text{for } 0 \le \frac{x}{L} \le 1 \tag{3}$$

where y_t is the throat width and L is the duct length. The non-dimensional parameter y_t/L (to be specified below) governs the "slenderness" of the duct.

Use a finite-volume formulation where flux balances are made in a Cartesian reference frame. Use the explicit MacCormack method where the alternating bias on the fluxes is achieved by using 0^{th} order interpolation in order to compute the fluxes on the cell faces. In the predictor step the fluxes at faces $i + 1/2$ and $j + 1/2$ are computed from the flow variables contained in the cells $i+1$ and $j+1$ respectively; in the corrector step these same fluxes are computed from the flow variables contained in the cells i and j respectively. Use inviscid wall boundary conditions on the duct walls and characteristic boundary conditions at the exit.

The supersonic duct is to have an inlet Mach number of $M_\infty = 2.3$ and an exit to inlet pressure ratio of 9.0. Initialize the problem with the isentropic steady quasi-one-dimensional solution for isentropic flow through the duct. This initial condition will not satisfy the given downstream boundary condition, and this mismatch should propagate upstream as a shock wave, which will become stationary when a steady-state is achieved.

Due to symmetry the flow is identical on the upper and lower halves. Thus, in the interest of efficiency, it makes sense to compute only the upper half of the duct. A symmetry (no flux) boundary condition can be applied along the x-axis.

In order to validate your code, run cases on meshes containing 50X16, 100X32, 200X64, 400X128, and 800X256 (Nx X Ny) points for a slenderness ratio of $y_t/L =$

0.02, and compare the computed results with the the exact quasi-1D solution. The solutions will not agree exactly even for a large number of mesh points since the quasi-1D solution neglects the transport in the y-direction. The comparison can be made reasonably close, however, by averaging your two-dimensional solution over y and comparing these averages with the quasi-1D result. Make a convergence plot where the rms difference between the computed and quasi-1D pressure is plotted as a function of the number of mesh points used in x (i.e. Nx). Plot the y-averaged pressure as a function of streamwise distance for each case and compare this with the exact quasi-1D result. Run additional cases for $y_t/L = 0.04$ and $y_t/L = 0.05$ on a 400X128 mesh and plot the resulting pressure distributions along with the quasi-1D result as in the other cases.

MacCormack's method is not quite stable for these cases when a skewed mesh is used. It is thus necessary to add a small amount of artificial dissipation. Ideally, one wants to set the dissipation coefficient just high enough to stabilize the solution. Excessive dissipation will degrade the solution in a non-physical manner. Try a dissipation coefficient of 0.005 for the $y_t/L = 0.02$ cases, 0.01 for the $y_t/L = 0.04$ case, and 0.015 for the $y_t/L = 0.05$ case. You may also find that lowering the CFL number may help in controlling instabilities. You should not need to lower the CFL number below about 0.7 however.

You will need a computational mesh to solve the various cases listed above. A code is available on the course website which will generate the meshes. You can also download mesh files directly for the required cases.

1 Items Required in Your Report

As a bare minimum, your report must contain the following items

1. A series of plots where you show a comparison of your computed pressure with the quasi-1D result for a slenderness of $y_t/L = 0.02$, using 50X16, 100X32, 200X64, 400X128, and 800X256 mesh points. The pressure is to be averaged in y, and plotted as a function of x/L (this is done in the code template, and the average pressure is written to file avg_solution.dat). Plot each case on a separate graph and label each one clearly. The plots must be large enough to clearly show differences between the computed and quasi-1D solutions. Indicate the number of non-dimensional time units each case was run and specify whether you used one-dimensional or two-dimensional artificial dissipation, as well as the numerical value of the dissipation coefficient. If you used a large dissipation during the transient, but then restarted the run using a lower value, quote the lower value (although you may want to describe your procedure, including what value you chose for the dissipation coefficient during the transient).

2. A convergence plot where the rms difference between the computed pressure and the quasi-1D solution is plotted as a function of the number of mesh points in the x direction. Use a log-log scale and add reference lines as appropriate. Discuss the results. Is the convergence affected by artificial dissipation? By oscillations near the shock? By intrinsic differences between 2D flow and the 1D approximation?

3. Pressure plots as in item 1, but for $y_t/L = 0.04$, and $y_t/L = 0.05$, computed on a 400X128 point mesh. As before, indicate the duration of the simulation in non-dimensional time units and describe the details of the artificial dissipation used. Discuss any differences between the computed results and the quasi-1D solution.

4. A copy of your program. **Please email this to me instead of printing it out.** This will not only save paper, but will enable me to run your program if I am curious about any of your findings. **You must send me your program in order to get full credit for the project**.

5. Any notes or derivations that are useful to help in understanding the program. I will not look at this section very carefully and it will impact your project grade negligibly. It is more for your benefit in providing a record of what is in the program should you want to use or modify it at a later date.

2 Optional Items

You can achieve a perfect score on the project if you do a good job on the required items listed above. In case you are motivated to enlarge the report, you may want to consider the following items. If these extra items are reported in an efficient and logical manner, you can receive some replacement credit for required items that were not addressed correctly.

1. Line plots of any other aspects of the solutions which you find interesting. Don't just plot everything you can think of. If you include additional plots, discuss them in a coherent manner which adds some value to your report.

2. 2D contour plots of the solution. Same caveat as in item 1.

3. Brief summary of results for other slenderness ratios. You should be able to obtain solutions for y_t/L up to at least 0.075, although these may be more tricky to keep stable. Somewhere between $y_t/L = 0.075 - 0.080$, the shock system will transition to a single normal shock, which will move all the way forward to the upstream boundary. This is physically correct as the duct walls become sufficiently steep that oblique shocks are no longer possible.

4. Anything else you can think of that adds value and interest to the report. If you do this, be mindful that I reserve the right to subtract points for repetitive or meagerly discussed plots, tables, or any other "filler" stuff.