

# 16.901 Project #2

## Due Date: April 4, 2pm

### 1 Background

We will simulate the compressible inviscid flow through a varying-area duct in this project. The goal is to gain some understanding of the convergence behavior of Computational Fluid Dynamics algorithms applied to this problem for varying mesh size and under different flow conditions. In particular, we will use a first-order upwind spatial approximation with a Forward Euler timestep algorithm.

The cross-sectional area of the duct has a constant area section, a smooth converging section to a throat, a smooth diverging section to a final constant area section. The specific geometry is given by,

$$\begin{aligned} \text{For } 0 \leq x \leq x_1 : \quad A(x) &= A_0, \\ \text{For } x_1 \leq x \leq x_2 : \quad A(x) &= A_0 + \frac{1}{2}(A_1 - A_0) \{1 - \cos[\pi(x - x_1)/(x_2 - x_1)]\}, \\ \text{For } x_2 \leq x \leq x_3 : \quad A(x) &= A_1 + \frac{1}{2}(A_2 - A_1) \{1 - \cos[\pi(x - x_2)/(x_3 - x_2)]\}, \\ \text{For } x_3 \leq x \leq x_4 : \quad A(x) &= A_2. \end{aligned}$$

For the problem in this project, we will set:

$$\begin{aligned} x_1 &= 1 \text{ m} \\ x_2 &= 2 \text{ m} \\ x_3 &= 3 \text{ m} \\ x_4 &= 4 \text{ m} \\ A_0 &= 10 \text{ m}^2 \\ A_1 &= 8 \text{ m}^2 \\ A_2 &= 12 \text{ m}^2 \end{aligned}$$

The flow through the duct will be set by a pressure difference occurring from inlet to the outlet. Specifically, at the inlet, we will assume that the total pressure and the total temperature are equal to sea level atmospheric conditions:

$$\begin{aligned} P_{0inlet} &= 101,327 \text{ N/m}^2 \\ T_{0inlet} &= 288.15 \text{ K} \end{aligned}$$

At the outlet, we will solve the flow for three different static pressures specifically,

$$p_{outlet}/P_{0inlet} = 0.70, 0.90, \text{ and } 0.995.$$

The quasi-1D Euler equations can be written in a control volume form as,

$$\frac{d}{dt} \int_{x_L}^{x_R} UA \, dx + F(U(x_L)) A(x_L) - F(U(x_R)) A(x_R) = S \tag{1}$$

where  $U$  and  $F$  are,

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho E \end{pmatrix}, \quad F = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u H \end{pmatrix},$$

and  $S$  is a source term vector due to the area-variation,

$$S = \begin{pmatrix} 0 \\ \int_{x_L}^{x_R} p dA \\ 0 \end{pmatrix}.$$

As described in Homework #4,  $\rho$  is the density,  $u$  is the flow velocity,  $p$  is the pressure, and  $E$  is the total energy.  $H$  is the total enthalpy and is related to the previous quantities by,

$$H = E + \frac{p}{\rho}.$$

Note, the total energy ( $E$ ) and total enthalpy ( $H$ ) are related to the energy ( $e$ ) and enthalpy ( $h$ ) by the following,

$$\begin{aligned} E &= e + \frac{1}{2}u^2 \\ H &= h + \frac{1}{2}u^2 \end{aligned}$$

To close this set of equations, we will assume the working fluid is air and can be assumed to be an ideal, perfect gas. In this case, the state equations for the pressure and energy are,

$$p = \rho RT, \tag{2}$$

$$e = c_v T, \tag{3}$$

where  $R$  is the gas constant,  $T$  is the temperature, and  $c_v$  is the specific heat at constant volume. Another useful thermodynamic quantity is the specific heat at constant pressure,  $c_p$ , which is related to  $R$  and  $c_v$  by,

$$c_p - c_v = R.$$

Also, the ratio of specific heats  $\gamma = c_p/c_v$ . For air, we will assume that,

$$\gamma = 1.4 \quad R = 287.06 \text{ m}^2(\text{s}^2\text{K})^{-1}$$

As shown in Homework #4, the pressure can be related to the conservative state vector by,

$$p = (\gamma - 1) \left( \rho E - \frac{1}{2} \rho u^2 \right).$$

Finally, for a perfect gas, the speed of sound,  $c$  is given by,

$$c = \sqrt{\frac{\gamma p}{\rho}}.$$

## 1.1 Discrete Equations for Interior Cells

We begin by dividing the duct into  $N_x$  equal divisions (producing  $N_x$  cells). The unknowns will be the average-values of the state vector  $U$  in each cell, i.e. you will be solving for  $U_j$  for  $j = 1$  to  $N_x$ .

For every cell in the interior of the domain (i.e.  $j = 2$  to  $N_x - 1$ ), we will use a discrete form of Equation (1) to find a new value at every iteration. Specifically, the discretization which you need to implement for the project is,

$$\frac{U_j^{n+1} - U_j^n}{\Delta t^n} V_j + R_j^n = 0, \tag{4}$$

where  $R_j^n$  is the local residual vector for cell  $j$  at iteration  $n$ . This local residual is defined as,

$$R_j^n = \hat{F}(U_j^n, U_{j+1}^n) A_{j+\frac{1}{2}} - \hat{F}(U_{j-1}^n, U_j^n) A_{j-\frac{1}{2}} - S_j^n. \tag{5}$$

The function  $\hat{F}$  has been written already (as a Matlab<sup>®</sup> script called **Fupwind.m**) and is available on the course website as an additional attachment. This function, known as a flux function, takes the left and right state vector (and the value of  $\gamma$ ) at a face between two cells, and returns the flux which has been determined by properly upwinding the states to the face. The resulting difference approximation would be equivalent to a standard 1st order upwind discretization if the problem were just scalar convection instead of the Euler equations. NOTE: the order in which the states are given in the flux function input list is important. Do not reverse them!

The source term is approximated as,

$$S_j^n = \begin{pmatrix} 0 \\ p_j^n (A_{j+\frac{1}{2}} - A_{j-\frac{1}{2}}) \\ 0 \end{pmatrix}. \quad (6)$$

Also,  $V_j$  is the volume of cell  $j$  and should be approximated by,

$$V_j = \frac{1}{2}(A_{j-\frac{1}{2}} + A_{j+\frac{1}{2}})\Delta x.$$

## 1.2 Timestep Calculation

The timestep will be set with the following CFL condition,

$$\Delta t^n = CFL \min_{j=1}^{N_x} (\Delta t_j^n),$$

where,

$$\Delta t_j^n = \frac{\Delta x}{|u_j| + c_j}.$$

While you can experiment with the CFL number you use to determine the timestep, von Neumann stability analysis suggests that the  $CFL \leq 1$  to remain stable. My suggestion would be to use a  $CFL = 0.9$  for all of the simulations you will perform in this project.

## 1.3 Boundary Condition Implementation

At the inlet, we need to determine the state vector  $U_1$  by imposing boundary conditions. Specifically, we need to have three pieces of information that can be manipulated to uniquely determine the state vector (since the state vector has three components). We assume the inlet flow to be subsonic, as a result, only two pieces of information may be set while the third piece of information must come from the interior states. The two pieces of information we will set are the total pressure and total temperature at the inlet. From cell  $j = 2$ , we will use the velocity,  $u_2$ . Thus, these three pieces of information can be used to determine the state vector  $U_1$ , i.e.

$$U_1 = U(P_{0inlet}, T_{0inlet}, u_2)$$

Note: by this notation, it is meant that given the total pressure, total temperature, and velocity, the entire state vector can be determined.

At the outlet, we also will assume a subsonic flow. In this situation, we specify only one condition and take two pieces of information from the interior. For this assignment, we will set the static pressure and take the entropy,  $S$ , and  $J_+$  (known as Riemann invariant) from the interior. A definition of entropy for an ideal gas is,

$$S = \frac{p}{\rho^\gamma}.$$

The definition of  $J_+$  is,

$$J_+ = u + 2\frac{c}{\gamma - 1}.$$

Thus, to find the outlet state vector, we need to combine this information,

$$U_{N_x} = U(p_{outlet}, S_{N_x-1}, J_{+N_x-1})$$

$N_x$	$p_{outlet}/P_{0inlet}$		
	0.70	0.90	0.995
20			
40			
80			
160			

Table 1: Iteration requirements versus grid size ( $N_x$ ) and pressure ratio.

## 1.4 Initial Condition

For all of the cases run, use an initial of stagnation flow from the inlet conditions. That is, set  $u = 0$ ,  $p = P_{0inlet}$ , and  $T = T_{0inlet}$  everywhere in the duct prior to the first iteration.

## 2 Assignment

### 2.1 Implementation of Discretization

Implement the above algorithm. A skeleton code has been attached to the 16.901 website which you may use to base your code on (it is called **Q1D.m**). Provide a source code listing attached **to the back of your report**.

### 2.2 Residual Convergence Check

Develop a residual measure which can be used to check the level of convergence to a steady state. As discussed in class, use a root-mean-square type measure of the residual which will approach an asymptotic answer as the grid is refined (i.e. the residual measure should not be strongly dependent on grid size). Describe your residual measure and demonstrate that it is fairly insensitive to grid size. Also, determine a reasonable tolerance on this measure which will be used to determine when a solution converges in the remaining study.

### 2.3 Simulations

Using the residual convergence check developed above, perform simulations for all three pressure ratios using for grids,  $N_x = 20, 40, 80,$  and  $160$ . For each pressure ratio, plot the Mach number distribution for each grid size (i.e. overlay the Mach number distribution from each grid). Also plot the total pressure ratio  $P_{0outlet}/P_{0inlet}$  versus pressure ratio  $p_{outlet}/P_{0inlet}$  for all four grids (overlay these four lines on a single plot). Similarly, plot the mass flow at the outlet  $\dot{m}$  versus pressure ratio  $p_{outlet}/P_{0inlet}$  for all four grids. Do the simulations appear to be asymptoting to a 'grid independent' solution?

### 2.4 Iterations and Timings

For the 12 simulations run in the previous section, provide tables of the number of iterations required to reach convergence and the CPU timings. Sample tables are given in Tables 1 and 2. Describe the trends observed versus both with respect to the grid size and the flow conditions. In particular, try to connect what you observe in these results to theory (THIS DISCUSSION IS IMPORTANT IN ORDER TO RECEIVE FULL CREDIT).

	$P_{outlet}/P_{inlet}$		
$N_x$	0.70	0.90	0.995
20			
40			
80			
160			

Table 2: CPU time requirements versus grid size ( $N_x$ ) and pressure ratio.