Lecture 7

Conservation Laws and the Finite Volume Method

7.1 Conservation Laws

In most engineering applications, the physical system is governed by a set of conservation laws. For example, in gas dynamics, the conservation of mass, momentum, and energy are applied to the gas. These conservation laws are often written in integral form for a fixed physical domain. Suppose we have a two-dimensional physical domain, Ω, with the boundary of the domain, $\delta\Omega$. Then, the canonical conservation equation assuming that the physical domain is fixed is of the form,

$$\frac{d}{dt}\int_{\Omega} U\,dA + \int_{\delta\Omega}\left[F(U)\vec{i} + G(U)\vec{j}\right]\cdot\vec{n}\,ds = \int_{\Omega} S(U,t)\,dA, \tag{7.1}$$

where U is the conserved state, F and G are the flux of the conserved state in the x and y directions, $\vec{n}$ is the outward pointing unit normal on the boundary of the domain, and S is a source term.

This conservation law can be written as a partial differential equation by applying the divergence theorem which states that,

$$\int_{\delta\Omega}\left[F\vec{i} + G\vec{j}\right]\cdot\vec{n}\,ds = \int_{\Omega}\left(\frac{\partial F}{\partial x} + \frac{\partial G}{\partial y}\right)\,dA. \tag{7.2}$$

Thus, Equation 7.1 becomes,

$$\frac{d}{dt}\int_{\Omega} U\,dA + \int_{\delta\Omega}\left(F\vec{i} + G\vec{j}\right)\cdot\vec{n}\,ds = \int_{\Omega} S\,dA,$$

$$\int_{\Omega}\frac{\partial U}{\partial t}\,dA + \int_{\Omega}\left(\frac{\partial F}{\partial x} + \frac{\partial G}{\partial y}\right)\,dA = \int_{\Omega} S\,dA,$$

$$\int_{\Omega}\left(\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} - S\right)\,dA = 0.$$

Thus, since this last equation would remain valid for any arbitrary domain, Ω, this means that the integrand must be zero everywhere, or, equivalently,

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = S. \tag{7.3}$$

Equation 7.3 is the conservation law written as a partial differential equation.

Example 7.1 (Conservation of Mass for a Compressible Fluid) *An example of a conservation law is the conservation of mass for a compressible fluid. Let the fluid density be $\rho(x, y, t)$ and the fluid x and y velocity components be $u(x, y, t)$ and $v(x, y, t)$, respectively. Then, the conservation of mass for the fluid is,*

$$\frac{d}{dt}\int_\Omega \rho \, dA + \int_{\delta\Omega} \left[\rho u \vec{i} + \rho v \vec{j}\right] \cdot \vec{n} \, ds = 0.$$

In terms of the canonical form,

$$\begin{aligned} U &= \rho, \\ F &= \rho u, \\ G &= \rho v, \\ S &= 0. \end{aligned}$$

Example 7.2 (The Euler Equations for a Compressible Fluid) *Often, multiple conservation laws are of interest. In this case, U is a vector of conserved states. Furthermore, F, G, and S are vectors. As an example, the Euler equations for a compressible fluid in two-dimensions are the combination of conservation of mass, x-momentum, y-momentum, and energy. In this case,*

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix} \qquad F = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uH \end{pmatrix} \qquad G = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho vH \end{pmatrix} \qquad S = 0.$$

The first row of these vectors represents the conservation of mass (see Example 7.1 for more). The second and third row represent conservation of x and y momentum, respectively. And, the fourth row represents conservation of energy. Note that in addition to the fluid density (ρ) and the x and y velocity components (u and v), the other quantities in these equations are:

$$\begin{aligned} p &= \textit{static pressure}, \\ E &= \textit{total energy}, \\ H &= \textit{total enthalpy}. \end{aligned}$$

The total energy and total enthalpy are related by,

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

This system of equations is not quite complete, however, since the number of equations does not equal the number of dependent variables in the equations. In particular, note that we have given five equations thus far (the four conservation equations and the relationship of H to E) while the number of dependent variables is six (i.e. ρ, u, v, p, E, and H). The final equation is an equation of state. Often, we assume an ideal gas and use the ideal gas law. In terms of the dependent variables we have introduced, the ideal gas law can be written as,

$$p = (\gamma - 1)\left[\rho E - \frac{1}{2}\rho(u^2 + v^2)\right], \tag{7.4}$$

where γ is the ratio of specific heats (for air, $\gamma \approx 1.4$). Many of you may be more familiar with the ideal gas law in the form, $p = \rho RT$ where R is the gas constant and T is the temperature. Equation 7.4 is (in fact) equivalent to $p = \rho RT$ but Equation 7.4 is used since $p = \rho RT$ introduces a new dependent variable (i.e. the temperature) and would therefore require yet another state equation to complete the system.

7.2 Convection

In many applications, especially those in fluid dynamics, convection is the dominant physical transport mechanism over much of the domain of interest. While diffusion is always present, often its affects are smaller except in limited regions (often near solid boundaries where boundary layers form due to the combined effects of diffusion and convection).

In this section, we will derive the convection equation using the conservation law as given in Equation 7.1. Specifically, let U be the 'conserved' scalar quantity and let the fluxes be given by,

$$F = uU, \qquad G = vU, \qquad S = 0, \tag{7.5}$$

where $u(x, y, t)$ and $v(x, y, t)$ are known velocity components. Note, a non-zero source term could be included, but for simplicity is assumed to be zero. As a PDE, this scalar conservation law is,

$$\frac{\partial U}{\partial t} + \frac{\partial}{\partial x}(uU) + \frac{\partial}{\partial y}(vU) = 0.$$

This equation can be manipulated by expanding the x and y derivatives into,

$$\frac{\partial U}{\partial t} + u\frac{\partial U}{\partial x} + v\frac{\partial U}{\partial y} = -U\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y}\right).$$

Often a reasonable assumption is that the velocity field is divergence free such that $\partial u/\partial x + \partial v/\partial y = 0$. In this case, we arrive at what is commonly referred to as the convection equation,

$$\frac{\partial U}{\partial t} + u\frac{\partial U}{\partial x} + v\frac{\partial U}{\partial y} = 0. \tag{7.6}$$

Physically, this equation states that following along the streamwise direction (i.e. convecting with the velocity), the quantity U does not change.

In developing numerical methods for convection-dominated problems, we will often rely on insight that can be gained from the convection equation for the specific case when the

velocity field is a constant value, i.e. $u(x, y, t) = u$ and $v(x, y, t) = v$. In this situation, the solution to Equation 7.6 has the following form,

$$U(x, y, t) = U_0(\xi, \eta) \quad \text{where} \quad \xi = x - ut, \quad \eta = y - vt, \tag{7.7}$$

where $U_0(x, y)$ is the distribution of U at time $t = 0$. By substitution, we can confirm that this indeed is a solution of Equation 7.6,

$$\begin{aligned} \frac{\partial U}{\partial t} + u\frac{\partial U}{\partial x} + v\frac{\partial U}{\partial y} &= \frac{\partial}{\partial t}U_0(\xi, \eta) + u\frac{\partial}{\partial x}U_0(\xi, \eta) + v\frac{\partial}{\partial y}U_0(\xi, \eta) \\ &= \frac{\partial U_0}{\partial \xi}\frac{\partial \xi}{\partial t} + \frac{\partial U_0}{\partial \eta}\frac{\partial \eta}{\partial t} + u\left(\frac{\partial U_0}{\partial \xi}\frac{\partial \xi}{\partial x} + \frac{\partial U_0}{\partial \eta}\frac{\partial \eta}{\partial x}\right) + v\left(\frac{\partial U_0}{\partial \xi}\frac{\partial \xi}{\partial y} + \frac{\partial U_0}{\partial \eta}\frac{\partial \eta}{\partial y}\right) \end{aligned}$$

The partial derivatives of ξ and η are,

$$\frac{\partial \xi}{\partial t} = -u, \quad \frac{\partial \xi}{\partial x} = 1, \quad \frac{\partial \xi}{\partial y} = 0,$$

$$\frac{\partial \eta}{\partial t} = -v, \quad \frac{\partial \eta}{\partial x} = 0, \quad \frac{\partial \eta}{\partial y} = 1.$$

Upon substitution of these partial derivatives,

$$\begin{aligned} \frac{\partial U}{\partial t} + u\frac{\partial U}{\partial x} + v\frac{\partial U}{\partial y} &= -u\frac{\partial U_0}{\partial \xi} + -v\frac{\partial U_0}{\partial \eta} + u\frac{\partial U_0}{\partial \xi} + v\frac{\partial U_0}{\partial \eta}, \\ &= 0. \end{aligned}$$

Thus, $U(x, y, t) = U_0(x - ut, y - vt)$ is a solution to the convection equation.

Example 7.3 (Two-dimensional Convection) *To illustrate the behavior of the 2-D convection equation, we consider a specific problem. Suppose that the velocity is at 45 degrees with respect to the x-axis such that,*

$$u = 1, \qquad v = 1.$$

Also, suppose that the initial distribution of U is,

$$U_0(x, y) = e^{-x^2 - 20y^2}.$$

In Figure 7.1, the initial distribution of U is shown (i.e. at $t = 0$) as well as the distribution of U at $t = 1$. Clearly, the contours have moved along the 45 degree line (shown as a dashed line in the figure). Initially, the contours are centered at the origin and then at $t = 1$ they are centered at $x = y = 1$. A plot of U along the dashed line is shown in Figure 7.2. Since this line is tangent to the convection direction, the distribution of U convects without changing shape as t increases.

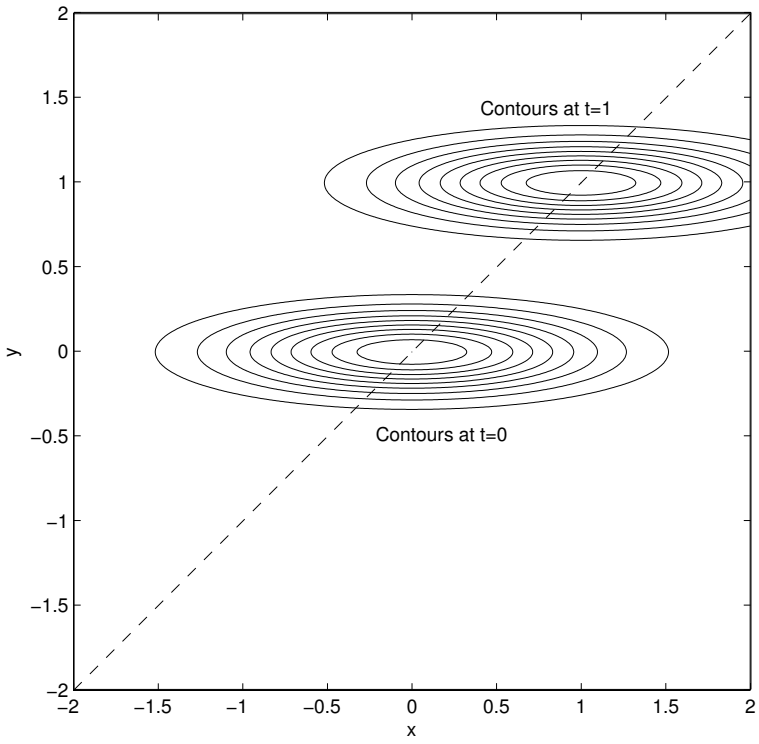

Figure 7.1: Distribution of U at $t = 0$ and at $t = 1$ for a convection problem with velocities $u = v = 1$.

7.3 Finite Volume Method for Convection

In this section, we will discuss the finite volume method. Our initial focus will be on convection and we will assume that the velocity field is divergence free. Thus, the integral conservation law (i.e. Equation 7.1) with fluxes given by Equation 7.5 is completely equivalent to the PDE for convection (i.e. Equation 7.6).

7.3.1 One-Dimensional Convection

In one-dimensional problems, the assumption that the velocity is divergence free, i.e. $\partial u/\partial x = 0$, forces the velocity to be constant with respect to x (though u could change with t).

The basis of the finite volume method is the integral convervation law. The essential idea is to divide the domain into many control volumes and approximate the integral conservation law on each of the control volumes. For example, as shown in Figure 7.3, cell i lies between the points at $x_{i-\frac{1}{2}}$ and $x_{i+\frac{1}{2}}$. Note that the points do not have to be equally-spaced.

The one-dimensional form of Equation 7.1 is

$$\frac{d}{dt}\int_{x_L}^{x_R} U\,dx + F(U)|_{x_R} - F(U)|_{x_L} = \int_{x_L}^{x_R} S(U,t)\,dx. \tag{7.8}$$

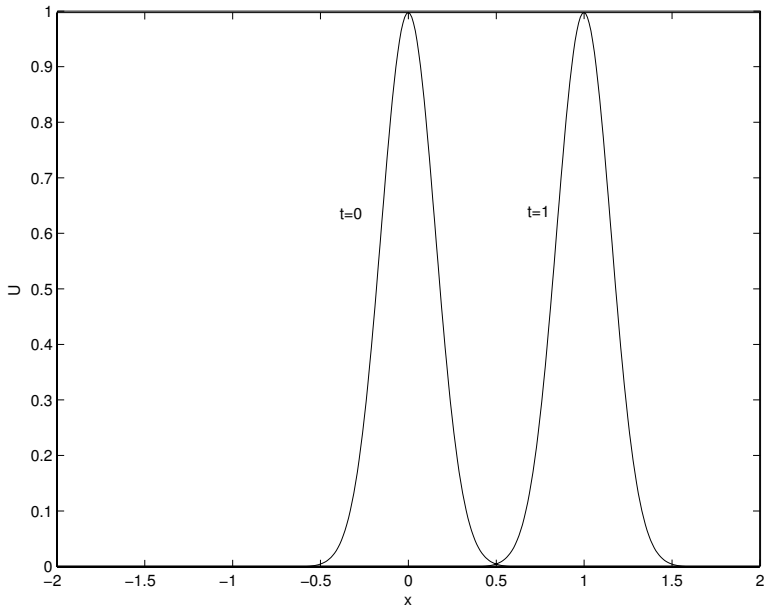

Figure 7.2: Distribution of U along $y = x$ at $t = 0$ and at $t = 1$ for a convection problem with velocities $u = v = 1$.

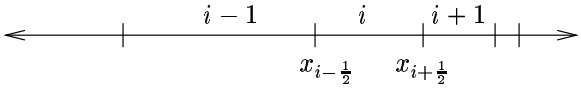

Figure 7.3: Mesh and notation for one-dimensional finite volume method.

Thus, applying this to control volume i (and recalling that $S = 0$ for convection) gives,

$$\frac{d}{dt}\int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} U\,dx + F(U)|_{x_{i+\frac{1}{2}}} - F(U)|_{x_{i-\frac{1}{2}}} = 0. \tag{7.9}$$

Next, we define the mean value of U in control volume i as,

$$U_i \equiv \frac{1}{\Delta x_i}\int_{x_{i-\frac{1}{2}}}^{x_{i+\frac{1}{2}}} U\,dx, \quad \text{where} \quad \Delta x_i \equiv x_{i+\frac{1}{2}} - x_{i-\frac{1}{2}}.$$

Then, Equation 7.9 becomes,

$$\Delta x_i \frac{dU_i}{dt} + F(U)|_{x_{i+\frac{1}{2}}} - F(U)|_{x_{i-\frac{1}{2}}} = 0. \tag{7.10}$$

At this point, no approximations have been made thus Equation 7.10 is exact. Now, we make the first approximation. Specifically, we assume that the solution in each control volume is constant,

$$U(x,t) = U_i(t) \quad \text{for} \quad x_{i-\frac{1}{2}} < x < x_{i+\frac{1}{2}}.$$

Thus, the finite volume approximation will be piecewise constant as shown in Figure 7.4.

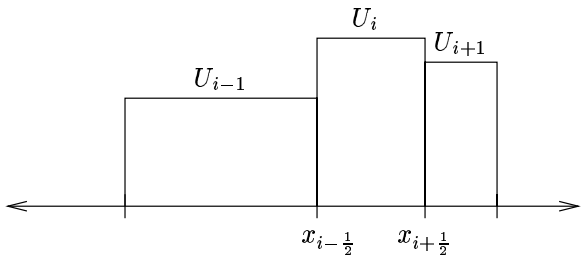

Figure 7.4: Piecewise constant solution for one-dimensional finite volume method.

With this assumed form of the solution, the next issue is to determine the flux at $i \pm \frac{1}{2}$ at a time t. This can be done with the knowledge that the solution convects with the velocity $u(t)$. Thus, for the initial instant after t (which we denote as $t^+ = t + \epsilon$ where ϵ is an infinitessimal, positive number):

$$U(x_{i+\frac{1}{2}}, t^+) = \begin{cases} U_i(t) & \text{if } u(t) > 0 \\ U_{i+1}(t) & \text{if } u(t) < 0 \end{cases}$$

The flux can be calculated directly from this value of U,

$$F(x_{i+\frac{1}{2}}, t^+) = \begin{cases} u(t)U_i(t) & \text{if } u(t) > 0 \\ u(t)U_{i+1}(t) & \text{if } u(t) < 0 \end{cases}$$

An alternative way to write this flux which is valid regardless of the sign of $u(t)$ is,

$$F(x_{i+\frac{1}{2}}, t^+) = \frac{1}{2}u(t)\left[U_{i+1}(t) + U_i(t)\right] - \frac{1}{2}|u(t)|\left[U_{i+1}(t) - U_i(t)\right]. \tag{7.11}$$

These fluxes, which use the upstream value of U to determine the flux. are known as an 'upwind' flux.

The final step in arriving at a full-discrete approximation for one-dimensional convection is to discretize Equation 7.10 in time. This can be done choosing any of the ODE integration methods we studied previously. For simplicity, we choose the forward Euler method so that the final fully-discrete form of the finite volume method is,

$$\Delta x_i \frac{U_i^{n+1} - U_i^n}{\Delta t} + F_{i+\frac{1}{2}}^n - F_{i-\frac{1}{2}}^n = 0, \tag{7.12}$$

where we use the notation,

$$F_{i+\frac{1}{2}}^n = \frac{1}{2}u^n\left(U_{i+1}^n + U_i^n\right) - \frac{1}{2}|u^n|\left(U_{i+1}^n - U_i^n\right). \tag{7.13}$$

Example 7.4 (Finite Volume Method applied to 1-D Convection) *The following Matlab script solves the one-dimensional convection equation using the finite volume algorithm given by Equation 7.12 and 7.13. The problem is assumed to be periodic so that whatever leaves the domain at $x = x_R$ re-enters it at $x = x_L$.*

```
% Script: convect1d.m

clear all;

% Set-up grid
xL = -4;
xR =  4;
Nx = 40; % number of control volumes
x = linspace(xL,xR,Nx+1);

% Calculate midpoint values of x in each control volume
xmid = 0.5*(x(1:Nx) + x(2:Nx+1));

% Calculate cell size in control volumes (assumed equal)
dx = x(2) - x(1);

% Set velocity
u = 1;

% Set final time
tfinal = 1;

% Set timestep
CFL = 0.5;
dt = CFL*dx/abs(u);

% Set initial condition to U0 = exp(-x^2)
% Note: technically, we should average the initial
% distribution in each cell but I chose to just set
% the value of U in each control volume to the midpoint
% value of U0.

U = exp(-xmid.^2);
t = 0;

% Loop until t > tfinal
while (t < tfinal),

  Ubc = [U(Nx), U, U(1)]; % This enforces the periodic bc

  % Calculate the flux at each interface
  F =   0.5*    u *( Ubc(2:Nx+2) + Ubc(1:Nx+1)) ...
      - 0.5*abs(u)*( Ubc(2:Nx+2) - Ubc(1:Nx+1));
```

```
  % Calculate residual in each cell
  R = F(2:Nx+1) - F(1:Nx);

  % Forward Euler step
  U = U - (dt/dx)*R;

  % Increment time
  t = t + dt;

  % Plot current solution
  stairs(x,[U, U(Nx)]);
  axis([xL, xR, -0.5, 1.5]);
  grid on;
  drawnow;

end
```

7.3.2 Two-Dimensional Convection

The finite volume discretization can be extended to two-dimensional problems. Suppose the physical domain is divided into a set of triangular control volumes, as shown in Figure 7.5. Application of Equation 7.1 to control volume A gives,

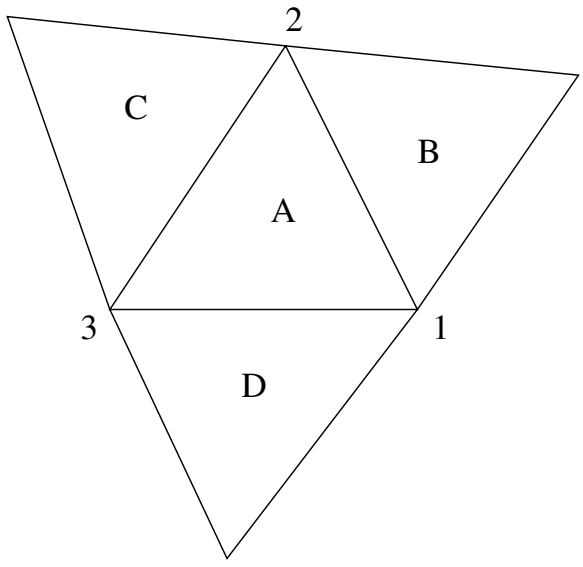

Figure 7.5: Triangular mesh and notation for finite volume method.

$$\frac{d}{dt}\int_{\Omega_A} U\, dA + \int_{\delta\Omega_A} H(U,\vec{n})\, ds = \int_{\Omega_A} S(U,t)\, dA, \tag{7.14}$$

where $H(U, \vec{n})$ is the flux normal to the face,

$$H(U, \vec{n}) \equiv \left[F(U)\vec{i} + G(U)\vec{j}\right] \cdot \vec{n}. \tag{7.15}$$

As in the one-dimensional case, we define the cell average,

$$U_A \equiv \frac{1}{A_A} \int_{\Omega_A} U \, dA,$$

where A_A is the area of control volume A. Thus, Equation 7.14 becomes,

$$A_A \frac{dU_A}{dt} + \int_{\delta\Omega_A} H(U, \vec{n}) \, ds = \int_{\Omega_A} S(U, t) \, dA.$$

In the case of convection, we again assume $S = 0$. Also, we expand the surface integral into the contributions for the three edges,

$$A_A \frac{dU_A}{dt} + \int_1^2 H(U, \vec{n}_{AB}) \, ds + \int_2^3 H(U, \vec{n}_{AC}) \, ds + \int_3^1 H(U, \vec{n}_{AD}) \, ds = 0,$$

where $\vec{n}_{AB}$ is the unit normal pointing from cell A to cell B, and similarly for $\vec{n}_{AC}$ and $\vec{n}_{AD}$.

As in one-dimensional case, we assume that the solution everywhere in the control volume is equal to the cell average value. Finally, the flux at each interface is determined by the 'upwind' value using the velocity component normal to the face. For example, at the interface between cell A and B,

$$H(U, \vec{n}_{AB}) \approx \hat{H}(U_A, U_B, \vec{n}_{AB}) \equiv \frac{1}{2} \vec{u}_{AB} \cdot \vec{n}_{AB} \left(U_B + U_A\right) - \frac{1}{2} |\vec{u}_{AB} \cdot \vec{n}_{AB}| \left(U_B - U_A\right), \tag{7.16}$$

where $\vec{u}_{AB}$ is the velocity between the control volumes. Thus, when $\vec{u}_{AB} \cdot \vec{n}_{AB} > 0$, the flux is determined by the state from cell A, i.e. U_A. Likewise, when $\vec{u}_{AB} \cdot \vec{n}_{AB} < 0$, the flux is determined by the state from cell B, i.e. U_B. The velocity, $\vec{u}_{AB}$ is usually approximated as the velocity at the midpoint of the edge (note: $\vec{u}$ can be a function of $\vec{x}$ in two-dimensions even though the velocity is assumed to be divergence free, i.e. $\partial u/\partial x + \partial v/\partial y = 0$). We use the notation $\hat{H}$ to indicate that the flux is an approximation to the true flux when $\vec{u}$ is not constant. Thus, the finite volume algorith prior to time discretization would be given by,

$$A_A \frac{dU_A}{dt} + \hat{H}(U_A, U_B, \vec{n}_{AB}) \Delta s_{AB} + \hat{H}(U_A, U_C, \vec{n}_{AC}) \Delta s_{AC} + \hat{H}(U_A, U_D, \vec{n}_{AD}) \Delta s_{AD} = 0.$$

The final step is to integrate in time. As in the one-dimensional case, we might use a forward Euler algorithm which would result in the final fully discrete finite volume method,

$$A_A \frac{U_A^{n+1} - U_A^n}{\Delta t} + \hat{H}(U_A^n, U_B^n, \vec{n}_{AB}) \Delta s_{AB} + \hat{H}(U_A^n, U_C^n, \vec{n}_{AC}) \Delta s_{AC} + \hat{H}(U_A^n, U_D^n, \vec{n}_{AD}) \Delta s_{AD} = 0. \tag{7.17}$$

Example 7.5 (Finite Volume Method for 2-D Convection on a Rectangular Mesh) *The following Matlab script solves the two-dimensional convection equation using a two-dimensional finite volume algorithm on rectangular cells. The algorithm is the extension of Equation 7.17 from triangular to rectangular cells. The problem is assumed to be periodic and have a constant velocity.*

```
% Script: convect2d.m

close all;
clear all;

% Specify x range and number of points
x0 = -2;
x1 =  2;
Nx = 40;

% Specify y range and number of points
y0 = -2;
y1 =  2;
Ny = 40;

% Construct mesh
x       = linspace(x0,x1,Nx+1);
y       = linspace(y0,y1,Ny+1);
[xg,yg] = ndgrid(x,y);

% Construct mesh needed for plotting
xp = zeros(4,Nx*Ny);
yp = zeros(4,Nx*Ny);
n = 0;
for j = 1:Ny,
  for i = 1:Nx,

    n = n + 1;
    xp(1,n) = x(i);
    yp(1,n) = y(j);

    xp(2,n) = x(i+1);
    yp(2,n) = y(j);

    xp(3,n) = x(i+1);
    yp(3,n) = y(j+1);

    xp(4,n) = x(i);
    yp(4,n) = y(j+1);

  end
end

% Calculate midpoint values in each control volume
```

```
xmid = 0.5*(x(1:Nx) + x(2:Nx+1));
ymid = 0.5*(y(1:Ny) + y(2:Ny+1));

[xmidg,ymidg] = ndgrid(xmid,ymid);

% Calculate cell size in control volumes (assumed equal)
dx = x(2) - x(1);
dy = y(2) - y(1);
A  = dx*dy;

% Set velocity
u = 1;
v = 1;

% Set final time
tfinal = 10;

% Set timestep
CFL = 1.0;
dt = CFL/(abs(u)/dx + abs(v)/dy);

% Set initial condition to U0 = exp(-x^2 - 20*y^2)
% Note: technically, we should average the initial
% distribution in each cell but I chose to just set
% the value of U in each control volume to the midpoint
% value of U0.
U = exp(-xmidg.^2 - 20*ymidg.^2);
t = 0;


% Loop until t > tfinal
while (t < tfinal),

  % The following implement the bc's by creating a larger array
  % for U and putting the appropriate values in the first and last
  % columns or rows to set the correct bc's
  Ubc(2:Nx+1,2:Ny+1) = U; % Copy U into Ubc
  Ubc(   1,2:Ny+1)   = U(Nx, :); % Periodic bc
  Ubc(Nx+2,2:Ny+1)   = U( 1, :); % Periodic bc
  Ubc(2:Nx+1,   1)   = U( :,Ny); % Periodic bc
  Ubc(2:Nx+1,Ny+2)   = U( :, 1); % Periodic bc

  % Calculate the flux at each interface
```

```
  % First the i interfaces
  F =   0.5*    u *( Ubc(2:Nx+2,2:Ny+1) + Ubc(1:Nx+1,2:Ny+1)) ...
      - 0.5*abs(u)*( Ubc(2:Nx+2,2:Ny+1) - Ubc(1:Nx+1,2:Ny+1));

  % Now the j interfaces
  G =   0.5*    v *( Ubc(2:Nx+1,2:Ny+2) + Ubc(2:Nx+1,1:Ny+1)) ...
      - 0.5*abs(v)*( Ubc(2:Nx+1,2:Ny+2) - Ubc(2:Nx+1,1:Ny+1));

  % Add contributions to residuals from fluxes
  R = (F(2:Nx+1,:) - F(1:Nx,:))*dy + (G(:,2:Ny+1) - G(:,1:Ny))*dx;

  % Forward Euler step
  U = U - (dt/A)*R;

  % Increment time
  t = t + dt;

  % Plot current solution
  Up = reshape(U,1,Nx*Ny);
  clf;
  [Hp] = patch(xp,yp,Up);
  set(Hp,'EdgeAlpha',0);
  axis('equal');
  caxis([0,1]);
  colorbar;
  drawnow;

end
```

7.4 Extensions of the Finite Volume Method

7.4.1 Nonlinear Systems

The basic finite volume approach can be extended to nonlinear systems of equations such as the Euler equations (see Example 7.2). The main issue in this extension is how to calculate an upwind flux when there is a system of equations. In one dimension, the basic finite volume discretization remains the same as given by Equation 7.13,

$$\Delta x_i \frac{U_i^{n+1} - U_i^n}{\Delta t} + F_{i+\frac{1}{2}}^n - F_{i-\frac{1}{2}}^n = 0.$$

The flux, however, must upwind (to some degree) all of the states in the equation. One relatively simple way in which this can be done is using what is known as the local Lax-

Friedrichs flux. In this case, the flux is given by,

$$F_{i+\frac{1}{2}}(U_i, U_{i+1}) = \frac{1}{2}\left[F(U_{i+1}) + F(U_i)\right] - \frac{1}{2}s_{\max}\left(U_{i+1} - U_i\right), \tag{7.18}$$

where $s_{\max}$ is the maximum speed of propagation of any small disturbance for either state U_i or U_{i+1}.

Example 7.6 (Lax-Friedrichs Flux for 1-D Euler Equations) *For the one-dimensional Euler equations, there are three equations which are approximated, i.e. conservation of mass, conservation of x-momentum, and conservation of energy. A small perturbation analysis can be performed which shows that the three speeds of propagation for this set of equations are* u, $u-a$, *and* $u+a$ *where* u *is the flow velocity and* a *is the speed of sound. Thus, the maximum speed will always be* $|u| + a$ *and the corresponding value of* $s_{\max}$ *for the Lax-Friedrichs flux is,*

$$s_{\max} = \max\left(|u|_i + a_i,\ |u|_{i+1} + a_{i+1}\right).$$

7.4.2 Higher-order Accuracy

The extension to higher-order accuracy will be discussed in class.