The pressure based and the density based solver

Boundary conditions

Dr. Gergely Kristóf
2008. 18-th April

Density based solver

Mainly for highly compressible (supersonic) flows

$$\frac{\partial \rho}{\partial t} + \nabla\cdot(\rho\,\vec{v}) = 0$$

$$\frac{\partial \rho u}{\partial t} + \nabla\cdot(\rho\,u\,\vec{v}) = -\frac{\partial p}{\partial x} + \nabla\cdot(\mu\nabla u) + \rho\,g_x$$

$$\frac{\partial \rho v}{\partial t} + \nabla\cdot(\rho\,v\,\vec{v}) = -\frac{\partial p}{\partial y} + \nabla\cdot(\mu\nabla v) + \rho\,g_y$$

$$\frac{\partial \rho w}{\partial t} + \nabla\cdot(\rho\,w\,\vec{v}) = -\frac{\partial p}{\partial z} + \nabla\cdot(\mu\nabla w) + \rho\,g_z$$

$$\frac{\partial \rho e}{\partial t} + \nabla\cdot(\rho\,e\,\vec{v}) = \nabla\cdot(-p\,\vec{v} + \underline{\tau}\cdot\vec{v}) + \nabla\cdot(\lambda\nabla T)$$

- Coupled iteration:

$$\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho e\end{bmatrix} \Rightarrow \begin{bmatrix}\rho\\ u\\ v\\ w\\ T\\ p\end{bmatrix}$$

$$e = c_v T + \frac{\vec{v}^2}{2}$$

$$p = \rho R T$$

Density models $\rho = f(T, p)$

Various forms of equation of state in FLUENT for single phase fluids:

- **Solid**: Constant
- **Liquid**: Constant; Boussinesq
- **Gas**: Constant → $\rho = \rho_0$; Ideal-gas (compressible) → $\rho = \frac{p}{RT}$; Incompressible-ideal-gas → $\rho = \frac{p_0}{RT}$; Boussinesq → $\rho = \rho_0 - \rho_0\beta(T - T_0)$

+ user defined density models:
  density (ρ) and the sound speed (a) must be defined. $a = \sqrt{\partial p/\partial\rho}\big|_{s=áll.}$

Material properties:
- Material Database;
- Material properties can be defined as temperature functions.

Incompressible vs. compressible

	Incompressible	Compressible
Density depend on p:	no	yes
Eq. of state:	ρ=áll. Boussinesq Inkomp. id. gáz	Ideal gas
Continuity:	replaced by the Poisson's eq. for p	solved for density in the original form
Total pressure:	$p_{tot} = p + \frac{\rho}{2}v^2$	$p_{tot} = p\left(1 + \frac{\kappa-1}{2}M^2\right)^{\frac{\kappa}{\kappa-1}}$
Time step size Courrant number	$\Delta t = C\frac{\Delta x}{v_\perp}\Big\|_{min}$	$\Delta t = C\frac{\Delta x}{a + \|\vec{v}\|}\Big\|_{min}$

Pressure based solver

Mainly for incompressible (or subsonic) flows

$$\nabla\cdot\vec{v} = 0$$

What's wrong with this ...

$$\frac{\partial u}{\partial t} = -\frac{\partial p/\rho_0}{\partial x} - \nabla\cdot(u\,\vec{v}) + \nabla\cdot(\nu\nabla u) + g_x = -\frac{\partial P}{\partial x} + G_x$$

$$\frac{\partial v}{\partial t} = -\frac{\partial p/\rho_0}{\partial y} - \nabla\cdot(v\,\vec{v}) + \nabla\cdot(\nu\nabla v) + g_y = -\frac{\partial P}{\partial y} + G_y$$

$$\frac{\partial w}{\partial t} = -\frac{\partial p/\rho_0}{\partial z} - \nabla\cdot(w\,\vec{v}) + \nabla\cdot(\nu\nabla w) + g_z = -\frac{\partial P}{\partial z} + G_z$$

By taking the divergence of the equation of motion and using the continuity equation we obtain a Poisson's equation for the pressure:

$$\Delta P = \nabla\cdot\vec{G}$$

- The continuity equation can be replaced by the Poisson's equation for the pressure (of pressure correction)
- Segregated iteration method ...

What do we mean by boundary conditions?

Mathematical interpretation

$$\frac{\partial}{\partial t}\int_V \rho\,\phi\,dV + \oint_A \rho\,\phi\,\vec{v}\cdot d\vec{A} = \oint_A (\vec{S}_A + \Gamma\nabla\phi)\cdot d\vec{A} + \int_V S_V\,dV$$

Fluxes and surface sources must be defined at the domain bounday.
The generic conservation equation in differential form:

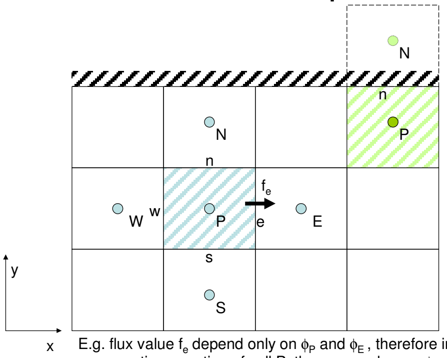
$$\frac{\partial \rho\phi}{\partial t} + \nabla\cdot(\rho\,\phi\,\vec{v}) = \nabla\cdot\vec{S}_A + \nabla\cdot(\Gamma\nabla\phi) + S_v$$

It is second order in space due to this term

Three types of boundary are possible for such PDEs:
1. BC of first kind: value of ϕ is given at the boundary;
2. BC of second kind: normal derivative of ϕ is given; $\frac{\partial\phi}{\partial n} = \nabla\phi\big|_n$
3. Mixed BC: linear combination of ϕ and it's normal derivative is given.

Boundary conditions for each conservation equations cannot be independently defined (E.g. we cannot use BC of 1-st kind for every velocity components along with BC of 1-st kind for pressure.)
Therefore FLUENT provides only boundary condition "packages" of well defined physical meaning.

Numerical interpretation



BC of 1-st kind:

$\phi_n = 0$:

$\phi_N = -\phi_P$

BC of 2-nd kind:

$\partial\phi_n / \partial y = 0$:

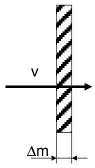
$\phi_N = \phi_P$

E.g. flux value f_e depend only on ϕ_P and ϕ_E , therefore in the discrete form of conservation equation of cell P, there are unknown ϕ values only from the neighboring cells (E,W,N,S).

Porous jump, porous zone

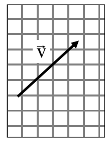
Porous jump: $$\Delta p = -\left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2\right) \Delta m$$

- Δp: pressure jump [Pa];
- Δm: thickness [m];
- α: face permeability [m²];
- C_2: inertial resistance;
- v: normal velocity.



Porous zone: $$S_i = -\left(\frac{\mu}{\alpha_i} v_i + C_{2,i} \frac{1}{2} \rho |v| v_i + C_0 |v|^{(C_1 - 1)} v_i\right)$$

S_i: i-th component of the volume force



Boundary condition packages

Inlet and outlet BCs

- Velocity-inlet
- Mass-flow-inlet
- Pressure-inlet
- Pressure-outlet
- Outflow
- Pressure-far-field
- Inlet-vent
- Intake-fan
- Outlet-vent
- Exhaust-fan
- Symmetry
- Wall
- Axis
- Periodic
- Interface

Discontinuity surfaces:

- o Interior
- o Porous-jump
- o Fan
- o Radiator
- o Wall

Volume sources:

- User defined source
- Porous zone
- Fixed value
- Rotating frame of reference

Boundary profiles

- Point profile (in ASCII file)
  - Simple to create (Excel editable)
  - You can Write and Read and Attach profiles.
- By User Defined Functions (written in C)
  - Profile can depend on any field variables (not only on spatial coordinates)
  - Program, interpret and hook.

Important notes

- Outflow cannot be used in the presence of Pressure Inlet or Pressure Outlet;
- Outflow cannot be used in compressible flow simulations;
- Back flow is not allowed through an Outflow (due to immediate convergence problems);
- Velocity Inlet provides unphysical results in compressible flow simulations (Mass Flow Inlet need to be used in these cases);
- Pressure Inlet is automatically changed to Pressure Outlet when back flow occurs (and the Pressure Outlet does similarly);
- There are three ways of branching the flow:
  - Outflow (with Flow Rate Weighting)
  - Multiple Pressure Outlets
  - Velocity Inlet with negative velocity (mathematically incorrect by works if proper care is taken)

Airfoil database

http://www.ae.uiuc.edu/m-selig/ads/coord_database.html