Numerical modeling of fluid flows

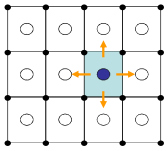
Department of Fluid Mechanics, BME

Lectures: Dr. Kristóf Gergely
Practices: Máté Lohász

April, 2010.

Finite volume method

Discrete solution



U: volume intensity of an arbitrary conserved quantity.

$$\frac{\partial}{\partial t}\int_V U\,dV + \oint_A \vec{F}\cdot d\vec{A} = \int_V S_V\,dV + \oint_A \vec{S}_A\cdot d\vec{A}$$

The conserved quantity per init mass of fluid:

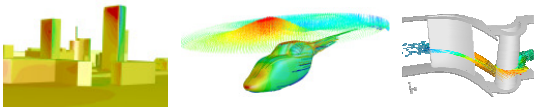
$$\Phi = U/\rho$$

Convective and conductive fluxes:

$$\vec{F}_C = \rho\Phi\bar{v} \qquad \vec{F}_D = -\Gamma\nabla\Phi$$

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

About the course



- Practices are in HSzK laboratories. Online tutorial: http://www.ara.bme.hu/~cfd
- Lectures:
  - Modeling theory;
  - Software basics;
  - Application examples.
- Practices: 5 tutorial example (with detailed instructions).
- Exam:
  - Practical: 1 simulation problem on the 6-th practice;
  - Theory: 1 midterm exam on the 7-th practice.

The generic transport equation in differential form:

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

conductive flux; convective flux

Transport equ.	ϕ
Continuity	1
x-momentum	u
y-momentum	v
z-momentum	w
Energy	e

Conservative form of the governing equations for single phase laminar flow:

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\bar{v}) = S_m$$

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

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

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

$\nabla\cdot(-p\underline{E} + \underline{\tau})$

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

Principles of CFD

- Our aim is the approximate solution of the governing PDE-s via numerical methods.
- Four major methods are in use:
  - finite difference method,
  - finite element method,
  - spectral methods,
  - finite volume method.
- In the field of CFD **finite volume** methods prevail.
- The domain is subdivided into smaller volumes (cells). This process is called: the grid generation or meshing.
- One single value of every field variable is stored in each cell center. Within the cell the field variables are approximated by linear functions.

Finite volume method

- The **integral form** of the above conservation equations are discretized.
- Divergence terms are converted into **surface integrals** over the facets enclosing the cells. The numeric approximation of the flux integral for one facet depend only on the unknown ϕ values stored in the centers of the two neighboring cells adjacent to the facet.
- As a result of this - so called discretization - process, every transport equation provides one (non-linear) **algebraic equation per cell**, e.g. if we have 5 transport equations and 1 000 000 cells then we obtain a system of 5 000 000 non-linear algebraic equations. In the case of time dependent problems we have to solve this system of equation in every time step.
- Each algebraic equations contain unknown ϕ values from one particular cell and from all neighboring cells. This is e.g. 5 unknowns per equations for tetrahedral grids.
- Due to the high number of unknowns and the non-linearity of the system of equations **iterative** solution methods have to be used. The solution is first **initialized**, and than, iteratively refined thus **converging** towards the final solution.
- Integral of fluxes over the boundary facets need to be defined in consistence with the physical characteristics of the region outside of the boundary by imposing additional mathematical conditions: **boundary conditions**.
- Surface integrals are numerically evaluated for every small facets connecting two neighboring cells. These values express the flow rates of conserved quantities (mass, momentum, energy). When we numerically calculate the integral of such conserved quantities for the whole domain, the surface integrals for internal facets are canceled, therefore the conservation equations for the whole domain are exactly fulfilled. E.g. the mass flow-rate through the inlet is equal to that of the outlet. This is called as the **conservative behavior** of the finite volume method.

Workflow

1.	Construction of the geometry	Design Modeller	ANSYS Workbench
2.	Meshing	Workbench Mesher	
3.	Marking boundary zones		
4.	Selection of the physical model and material properties	FLUENT	
5.	Boundary values		
6.	Numerical parameters		
7.	Initialization		
8.	Iteration		
9.	Evaluation of the results	CFD-Post	

This process will be demonstrated on a rounded orifice simulation example…