Computational Fluid Dynamics

Jathavan Sriram
Seminar Modeling and simulation
Department of Informatics
Scientific Computing (WR)
University of Hamburg
2013
Outline

- Introduction
- Models & Numerics
- Simulation
- Validation
- Conclusion
Introduction

Computational Fluid Dynamics ...

- Method of fluid dynamics
- Uses numerics
- Solve problems that involve fluid (flows)
Fluid flow problems

- **Science**
  - Weather forecast
  - Climate simulation
  - Medicine

- **Industry**
  - External flow (e.g. aerodynamics)
  - Internal flow (e.g. valve)
General types of CFD problems

<table>
<thead>
<tr>
<th>Problem</th>
<th>Given</th>
<th>Search</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>(Re)calculation</td>
<td>Geometry (Volume, Contour, Boundary,...)</td>
<td>Flow characteristics</td>
<td>Dam break</td>
</tr>
<tr>
<td>Construction</td>
<td>Flow characteristics</td>
<td>Geometry</td>
<td>Construction of oil pumps</td>
</tr>
<tr>
<td>Combined</td>
<td>Geometry &amp; Flow charac.</td>
<td>Optimum of both</td>
<td>Construction of airfoils</td>
</tr>
</tbody>
</table>
Models & Numerics

- Major model types
- Fluid classification
- Coupled PDE: Navier-Stokes equations
- ...in depth
## CFD Models

<table>
<thead>
<tr>
<th>Mesh based</th>
<th>Mesh free</th>
</tr>
</thead>
<tbody>
<tr>
<td>• More mathematically „correct“</td>
<td>• „Practical results“ not always accurate</td>
</tr>
<tr>
<td>• Discretization with FDM, FVM, FEM</td>
<td>• Discretization with placed particles: SPH</td>
</tr>
<tr>
<td></td>
<td>• Tracing particles</td>
</tr>
</tbody>
</table>
Mesh based models

- „Euler perspective“ – static reference system
- Finite Differences Method / Finite Volume Method
- Global, mostly static grids
- „Lagrangian perspective“ – Finite Element Method
- Structured or unstructured grids
- Global, often dynamic grids
Particle based model

- „Lagrangian perspective“ – dynamic reference system
- Smoothed Particle Hydrodynamics (SPH)
- Local, individual particles
- Each particle hold physical quantities like pressure, mass, density, ...
- Inherently takes care of conservation laws
Fluid classification (models itself!)

Incompressible (Liquid water)

Compressible (High velocity gas)

laminar

viscid

inviscid

newtonian

Fluids
Most relevant physical quantities

- Velocity field: $u$
- Pressure: $p$
- Density: $\rho$
Euler equations

- Describes flux in fluids
- No viscosity & No heat conduction
- Focus: Conservation of Momentum

\[ \frac{dv}{dt} + (v \cdot \nabla)v + \frac{1}{\rho} \nabla p = 0 \]
Navier-Stokes equations

- Time and space model for laminar, viscose flux of incompressible fluids
- With Viscosity $\rightarrow$ includes friction
- General description $\rightarrow$ need more equations!
## Classical Discretization methods

<table>
<thead>
<tr>
<th></th>
<th>Finite difference method (FDM)</th>
<th>Finite volume method (FVM)</th>
<th>Finite element method (FEM)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Pro</strong></td>
<td>Simple implementation</td>
<td>Conservation of quantities (flux, mass, energy, ...)</td>
<td>Very precise High Stability</td>
</tr>
<tr>
<td><strong>Con</strong></td>
<td>• Very slow computation</td>
<td>• Faster computation</td>
<td>• Mostly unstructured grids</td>
</tr>
<tr>
<td></td>
<td>• Structured/Unstructured (adaptive) grids</td>
<td>• High memory consumption</td>
<td>• High memory consumption</td>
</tr>
<tr>
<td></td>
<td>• Higher detail require denser mesh</td>
<td>• Higher detail require denser mesh</td>
<td>• Complex implementation</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• Complex mesh generation</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>• For strong deformations remeshing required</td>
</tr>
<tr>
<td><strong>Typical app</strong></td>
<td>Misc (Fluids, Solids, ...)</td>
<td>Fluids</td>
<td>Solids</td>
</tr>
</tbody>
</table>

- **FDM**: Finite difference method
- **FVM**: Finite volume method
- **FEM**: Finite element method
Steps to a CFD simulations

(Re)calculation Problem
- Given: Geometry volume
- Search: Flow characteristics

Relevant physical effects
- Selection of model equations
- Selection of initial values and/or
- Selection of boundary conditions

Selection and generation of problem geometry

Numerical method
- Selection of a viable numerical method that fits the problem
- Implementation

Analysis of results

Computation
Practical simulation

- Particle based
  - Smoothed-particle hydrodynamics
  - Product: „Realflow“ [S1]
  - Used in the vfx industry

- Grid based
  - Finite Volume Method
  - Free software: „OpenFoam“ [S2]
Validation

- In space (absence of certain forces)
- With real standard models
- Cross simulation comparison
- Prediction comparison (e.g. in weather forecast)
Take home message

- Classical physical model: Navier-Stokes equations
- FVM and FDM are standard
- Not one equation for all fluid problems
Thank you for your attention!
References

Literature

Figures
[F2] ebm-papst UK Ltd: Service Listing, CFD analysis for a backward curved centrifugal fan, URL: http://www.ebmpapst.co.uk/media/images/fans/Stromungsbild.jpg, United Kingdom, 2013

Software