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

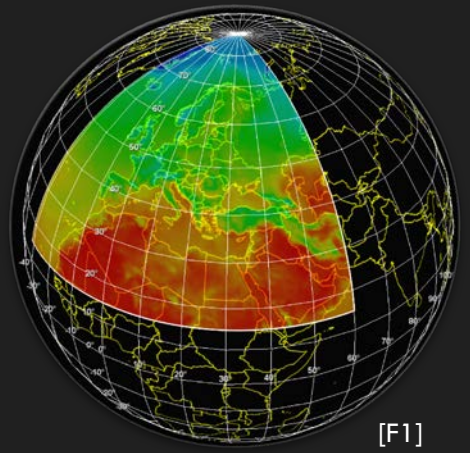
Introduction

Computational Fluid Dynamics ...

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

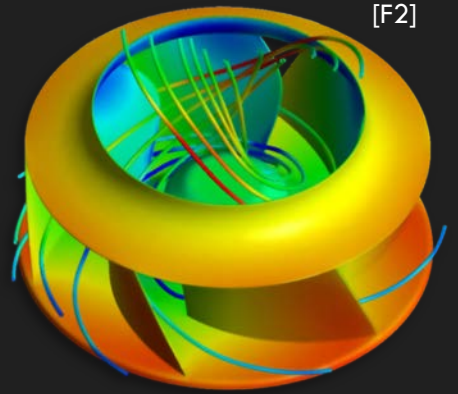
Fluid flow problems

- Science
  - Weather forecast
  - Climate simulation
  - Medicine
- Industry
  - External flow (e.g. aerodynamics)
  - Internal flow (e.g. valve)

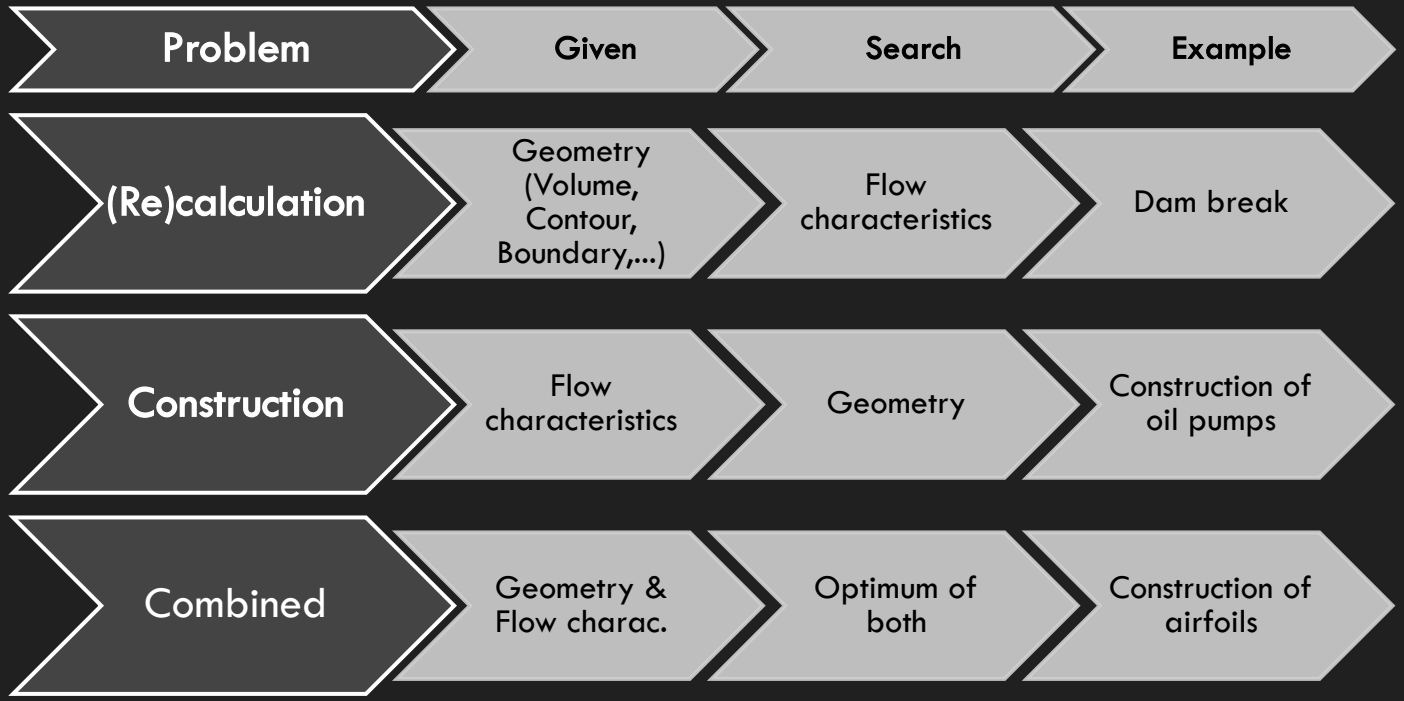


[F1]

[F2]



General types of CFD problems



Models & Numerics

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

CFD Models

Mesh based

- More mathematically„correct“
- Discretization with FDM, FVM, FEM

Mesh free

- „Practical results“ not always accurate
- Discretization with placed particles: SPH
- Tracing particles

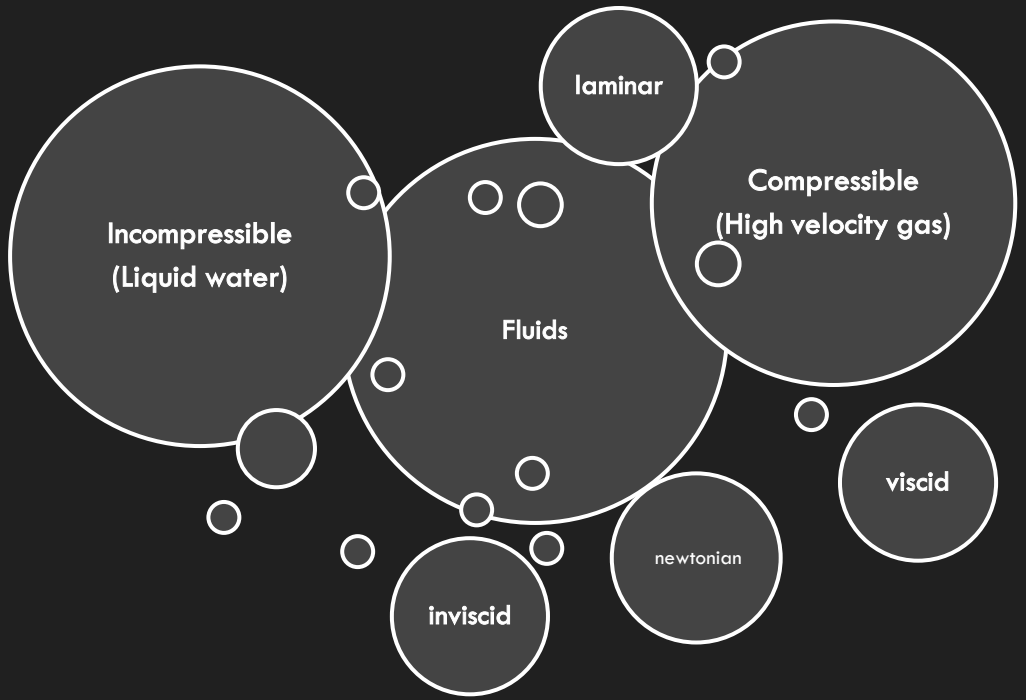
Mesh based models

- „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!)



Most relevant physical quantities

- Velocity field: u
- Pressure: p
- Density: ρ

Euler equations

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

$$\frac{dv}{dt} + (v \cdot \nabla)v + \frac{1}{\rho}\nabla p = 0$$

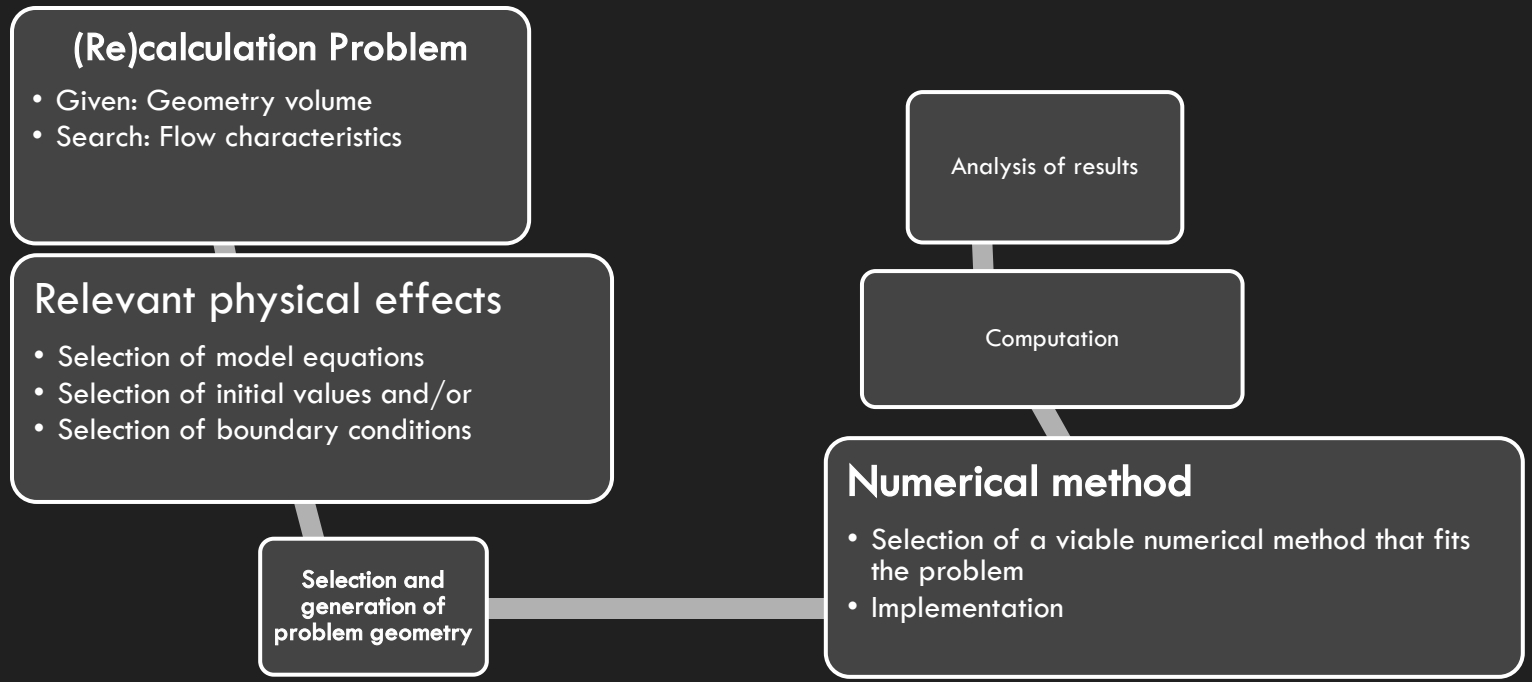
Navier-Stokes equations

- Time and space model for laminar, viscose flux of incompressible fluids
- With Viscosity → includes friction
- General description → need more equations!

Classical Discretization methods

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

Steps to a CFD simulations



Practical simulation

- Particle based
  - Smoothed-particle hydrodynamcis
  - 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

[L1] McGuffie K., Henderson-Sellers A.: A climate modelling primer,

Wiley&Sons, Chichester, England, 1997

[L2] Press William H.: Numerical Recipies 3rd edition,

Cambridge University Press, New York, USA, 2007

[L3] Ferziger J.H., Peric M.: Computational Methods for Fluid dynamics,

Springer Verlag, Berlin, Germany, 2002

Figures

[F1] Sriram J. , Sokol L. : Arjuna – A Java weather simulator, Screenshot,

, Project Scientific Computing, Germany, 2011

[F2] ebm-papst UK Ltd: Service Listing, CFD analysis for a backward curved centrifugal fan,

, URL: http://www.ebmpapst.co.uk/media/images/fans/Stroemungsbild.jpg, United Kingdom, 2013

Software

[S1] Next Limit Technology : Realfow 2012 , Fluid simulator,

, URL: http://www.realflow.com, Spain, 2013

[S2] OpenCFD Ltd. (ESI Group) : OpenFoam, Fluid simulator,

, URL: http://www.openfoam.com/, United Kingdom, 2013