Wind farm aerodynamics
Overview and plans

Benjamin Sanderse

March 6, 2009

Overview

- Background for current study
- Research goal
- Wind turbine and wind farm aerodynamics
- Engineering models
- CFD and status of current research
- Research directions

Background

- Reasons for grouping turbines in farms:
  - Lower construction and maintenance costs
  - Limited availability of off-shore sites
- Disadvantage:
  - Reduction in power production

- Numerical and experimental simulation of wind farm aerodynamics:
  - Understand physical phenomena
  - Optimize wind farms

Background

- Reasons for using CFD:
  - Find mutual influence of turbines in farms
  - Difficult situations: high rotor loading, yawed conditions, complex terrains
  - Model the atmosphere
  - Good quality experimental data is still limited. Optimization nearly impossible.

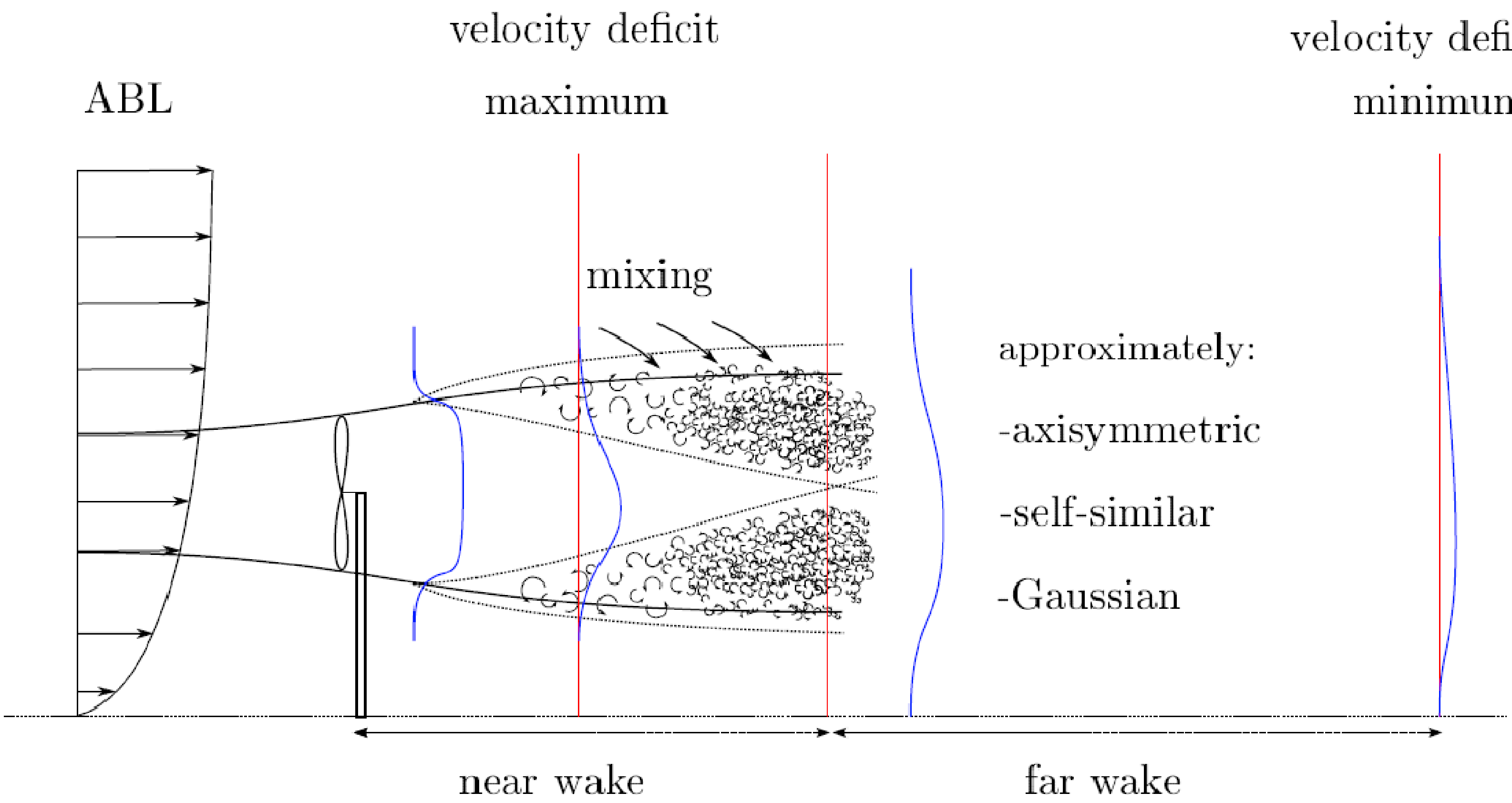
- Current ECN code (WAKEFARM) limited applicability

Research goal

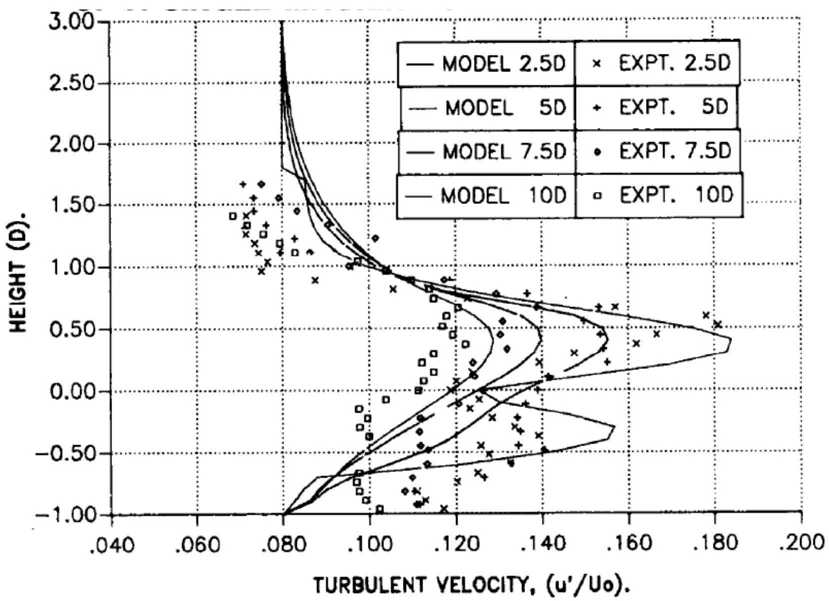
- **Simulation**:
  - Average velocity profiles for power production and farm efficiency
  - Turbulence fluctuations and intensity for blade loading
  - Wake meandering

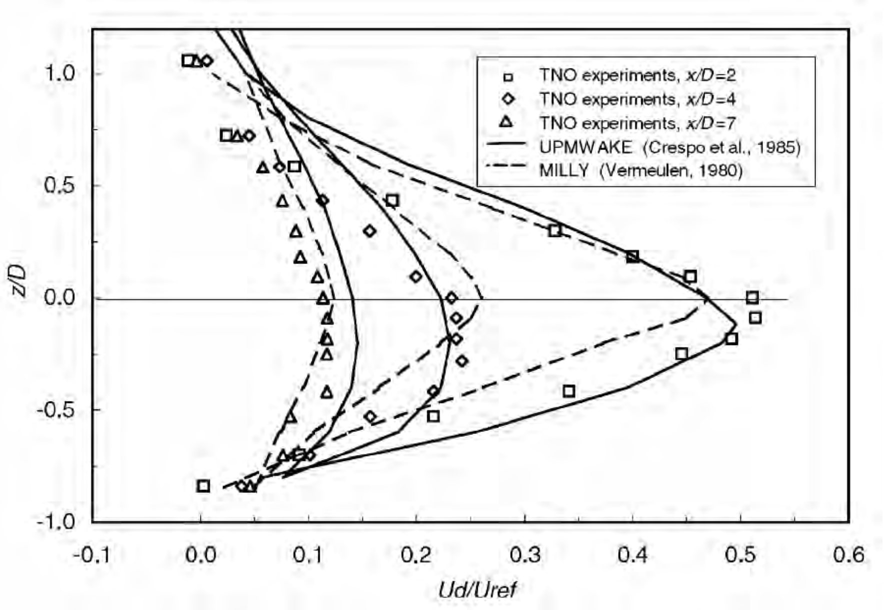
- **Optimization**

Wind turbine aerodynamics



Wind turbine aerodynamics



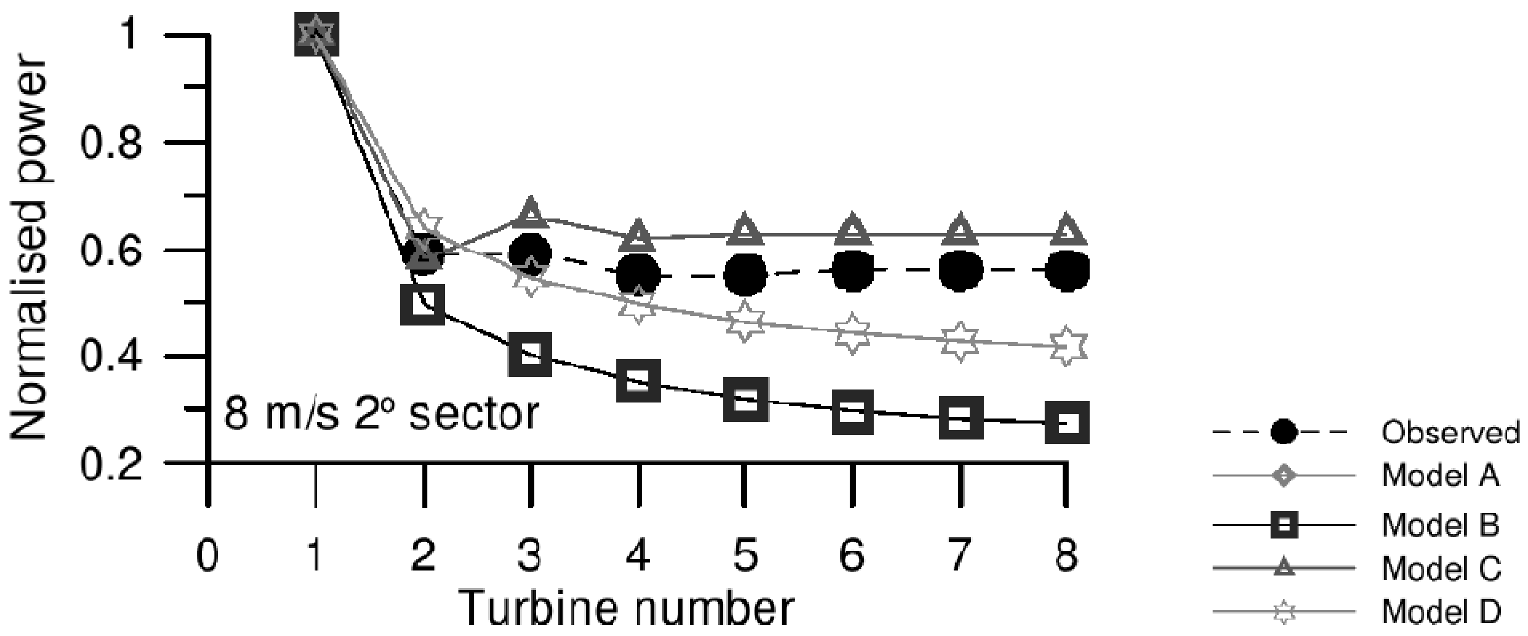


Wind farm aerodynamics

- Array efficiency:
  - Field geometry and terrain lay-out
  - Ambient turbulence (off/on-shore ...)
    - Mixing and wake recovery
  - Operating settings
    - Thrust coefficient
    - Yaw angle
    - …

Wind farm aerodynamics

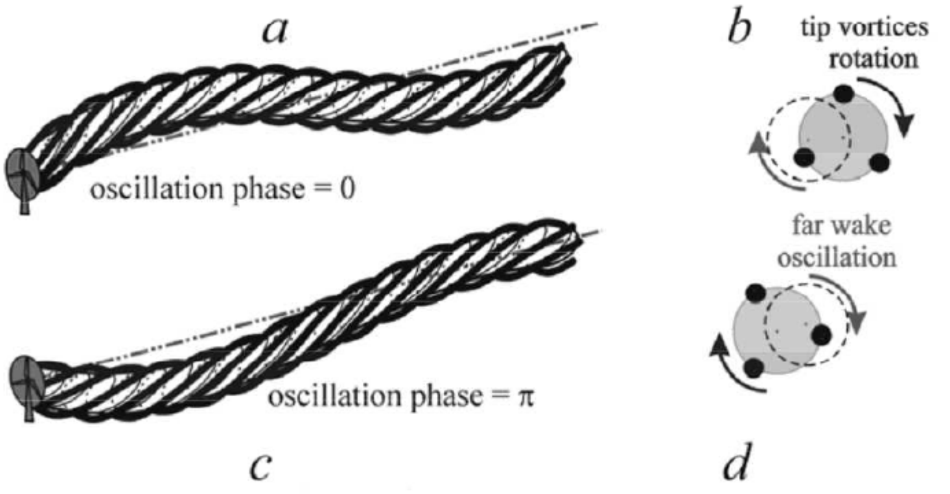
- Typical performance of downstream turbines



Example result from UPWIND Wp8 (EWEC 2007)

Wind farm aerodynamics

- Wake meandering
  - Variability in wind direction; large atmospheric eddies; passive tracers
  - Wake instability similar to bluff body aerodynamics



Larsen et al., Dynamic wake meandering modeling

Engineering models

- Rotor aerodynamics:
  - Blade element momentum
  - Lifting line
  - Panel method

- Wake aerodynamics:
  - Vortex wake methods
  - Wake and windfarm models:
    - Boundary layer models: distributed roughness elements
    - Kinematic models: self-similar profiles

Computational Fluid Dynamics

- Methods based on first principles: conservation of mass, momentum, energy

- Nature of the flow:
    - Unsteady: blade rotation, dynamic inflow, blade stall
    - Incompressible (?)
    - Viscous, turbulent

− n¢ I HL

H¢ / HP l »¢ n− d¢ I 1 f / G − ˘ l B− Q¢ S

Computational Fluid Dynamics

- Methods based on first principles: conservation of mass, momentum, energy

- Nature of the flow:
  - Unsteady: blade rotation, dynamic inflow, blade stall
  - Incompressible (?)
  - Viscous, turbulent

Challenges for CFD

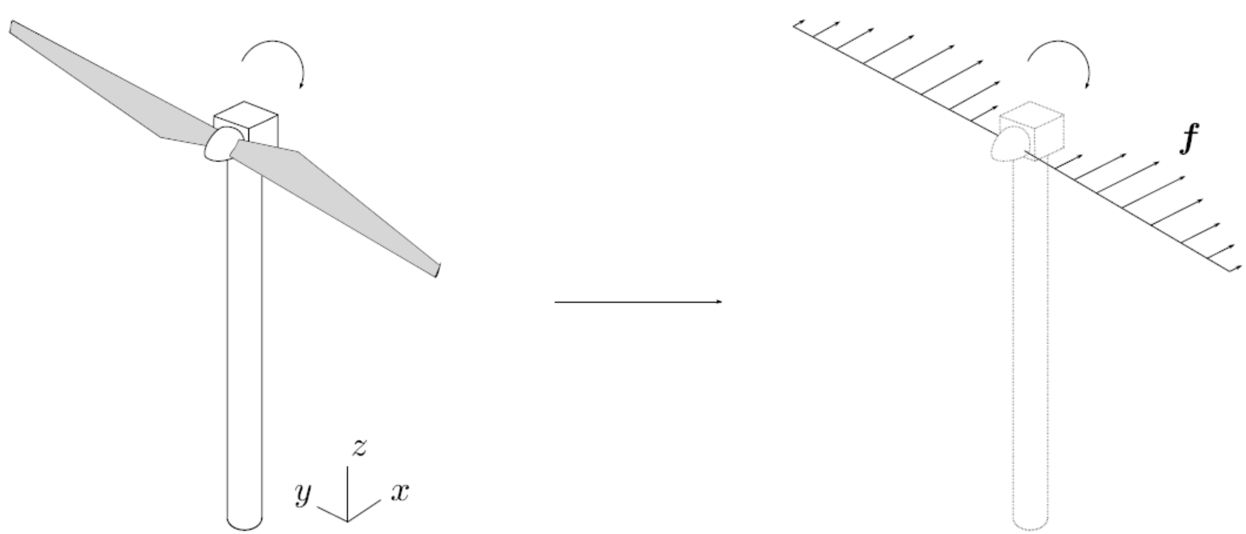
− n¢ I HL

H¢ / HP l »¢ n− d¢ I 1 f / G − ˘ l B− ǫ¢ S

CFD – rotor modeling

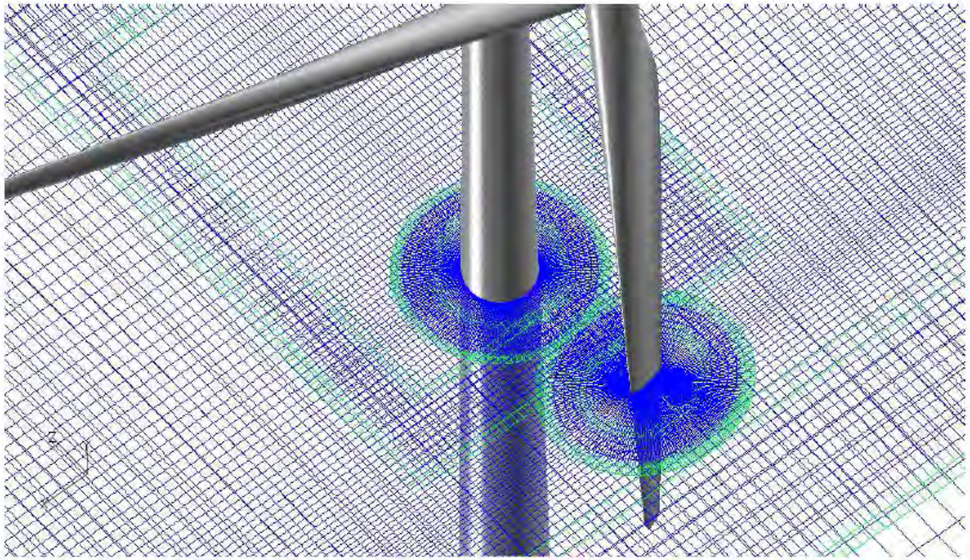
- Generalized actuator disk approach (similar to BEM)
  - Actuator disk/line/surface

H¢ l »¢ n⁻ d¢ I w l ƒ_ ˘ l B⁻ ϙ¢
HP G



CFD – rotor modeling

- Direct modeling
  - Chimera (overset) grids
  - Computationally intensive
  - Transition prediction very difficult



Zahle and Sorensen; Overset Grid Flow Simulation on a Modern Wind Turbine

CFD – wake modeling

- Parabolic
  - Fast solution by downstream marching
  - Pressure gradients and diffusive effects in axial direction neglected
  - Model for near wake necessary

CFD – wake modeling

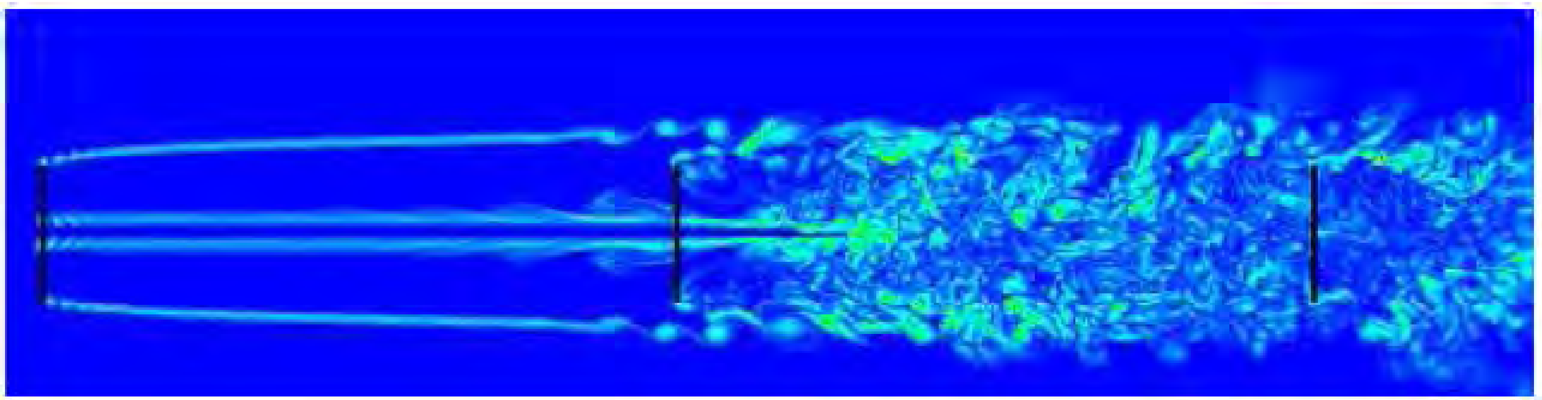
- Existing codes:
  - UPMWAKE (Crespo et al.):
    - Different turbulence models have been implemented, latest being LES
    - ECN: WAKEFARM (RANS)
  - Ainslie-type models, e.g. WindFarmer, WindPRO

CFD – wake modeling

- Elliptic
  - Entire flow domain
  - First mainly axisymmetric, steady, RANS; lately improvements to unsteady LES
  - Existing codes, e.g.:
    - DTU/Risø (Denmark)
    - CRES/NTUA (Greece)
    - CENER (Spain)
    - ETS (Canada)
    - CeSA(Portugal)

CFD – wake modeling

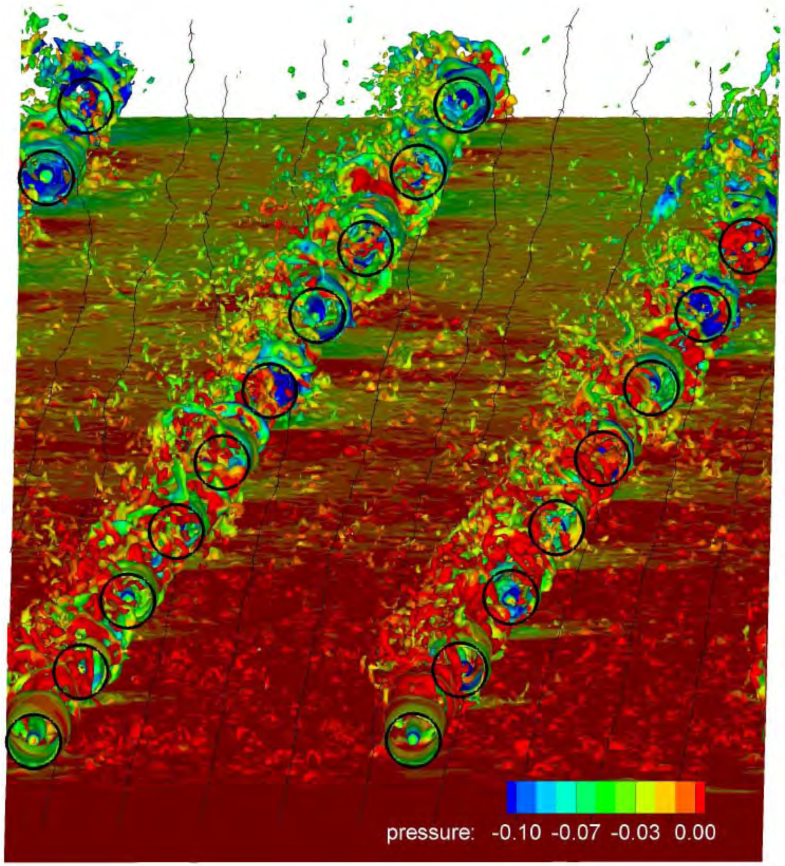
- Multiple wake simulation



N. Troldborg, Actuator line modeling of wind turbine wakes, 2008

CFD – wake modeling

- Wind farm simulations:
  - DTU/HG
  - CENER
  - CRES/NTUA



S. Ivanell, Numerical Computation of Wind Turbine Wakes, 2009.

CFD – helicopter wake modeling

- Tracking vortices:
  - Higher order ENO methods
  - Discontinuous Galerkin
  - Vorticity confinement
  - Lagrangian techniques

CFD – boundary conditions

- Inflow: turbulent atmospheric boundary layer
- Models:
  - Panofsky & Dutton
  - Mann: ‘turbulence generator’ with same statistics as atmosphere
  - Bechmann: LES simulation (with EllipSys3D)

CFD – validation

- Numerical data: benchmark tests
- Experimental data:
  - Wind tunnel tests, e.g. NREL and MEXICO
  - Field tests, e.g. UPWIND data sets, Tjaereborg, Sexbierum

Conclusions

- Requirements for a CFD code:
  - Incompressible (?)
  - Unsteady
  - Elliptic
  - Turbulence model, anisotropic: RSM/LES
  - Atmospheric inflow model
  - Implicit rotor modeling (?)
  - Earth-fixed reference frame

Discussion

Unanswered questions:

- What is the influence of the geometry of the blade on the wake?
- How much improvement do we expect with regard to current codes? How large is the margin?

Research directions

- Candidates:
  1. Actuator line/surface
  2. Immersed boundary method
  3. Vortex method

- Judging criteria
  - Computational effort vs. accuracy
  - Ease of mesh generation
  - Extension to future applications
  - Ease of dealing with complex inflow

1. Actuator line/surface

- ‘Classic’ method, used by different groups
- Blades represented by forces
- Fully conservative or ‘mimetic’ method
- Cartesian mesh with adaptive mesh refinement

Pro:

+ Mesh generation

+ Existing solvers and turbulence models

Con:

- Correctly simulating high Reynolds number flow, artificial diffusion

- Dependence on wind tunnel data or CFD results

- Future developments?

Mimetic methods

- Mimicking properties of differential operators, e.g.

  »¢ n− d I 1 »¢ n− d

- Symmetry-preserving:

  Convection: ' I 1 ' …

  Diffusion: [I […

- Stable on any grid. Conservation of mass, momentum, AND energy
- Staggered grids are a natural manner to achieve this

2. Immersed boundary method

- Blades ‘swim’ through mesh
- Fully conservative or ‘mimetic’ method
- Cartesian mesh with adaptive mesh refinement

Pro:

+ Mesh generation

+ Existing solvers and turbulence models

+ Physically sound

+ Attractive for future work:

  - fine mesh can resolve boundary layers
  - aeroelastic applications

+ Yawed flow

Con:

- Correctly simulating high Reynolds number flow

-Mesh fineness near boundary

-Time consuming?

2. Immersed boundary method

- Blades ‘swim’ through mesh
- Fully conservative or ‘mimetic’ method
- Cartesian mesh with adaptive mesh refinement

Pro:

+ Mesh generation

+ Existing solvers and turbulence models

+ Physically sound

+ Attractive for future work:

    - fine mesh can resolve boundary layers
    - aeroelastic applications

+ Yawed inflow

Con:

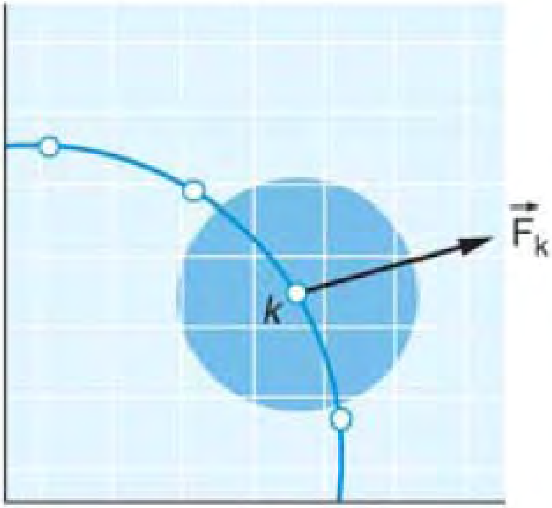
- Correctly simulating high Reynolds number flow
- Mesh fineness near boundary
- Time consuming?

Can a (coarse grid) IBM be competitive with a generalized actuator disk method?

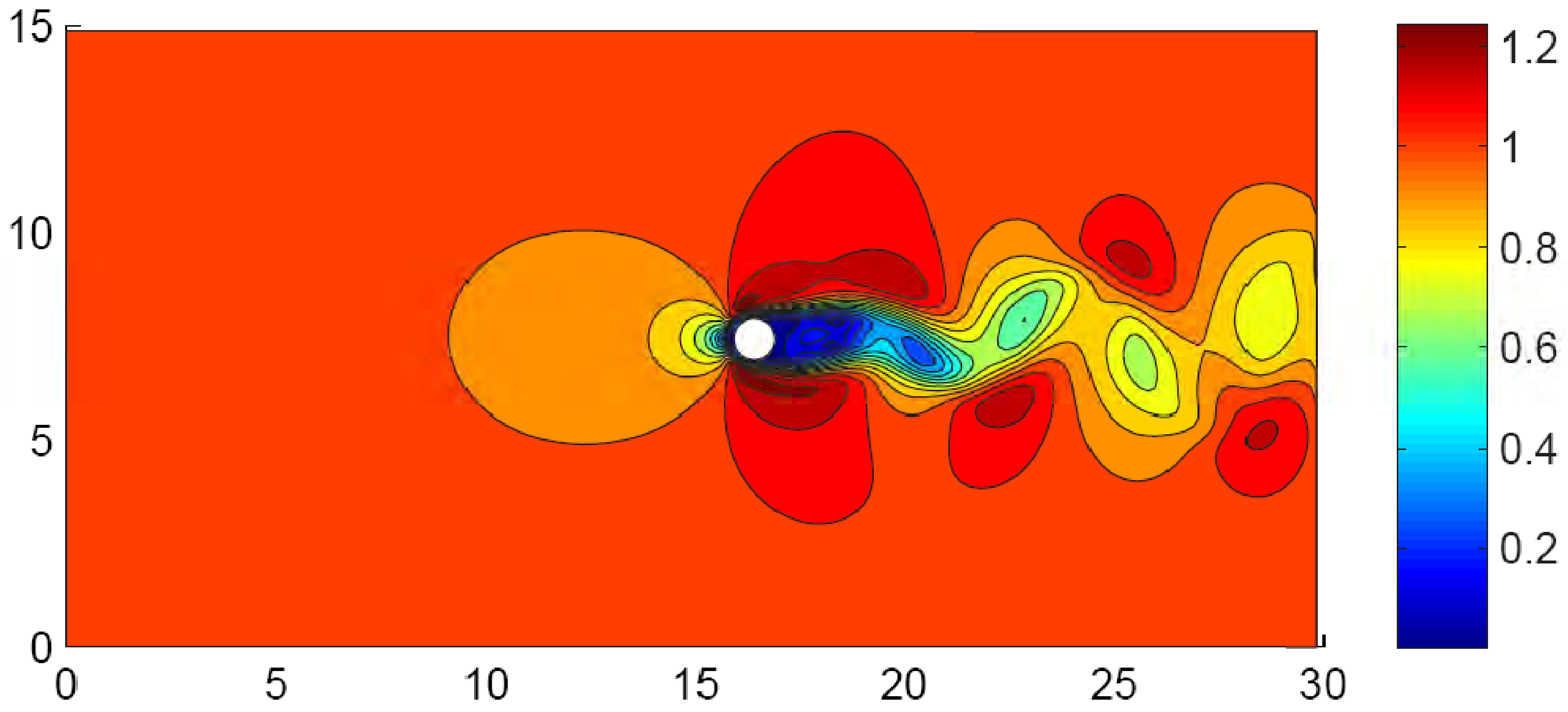
2. Immersed boundary method

- Continuous forcing:
  - Moving bodies
  - Aeroelastic computations
  - Smearing due to transfer Lagrangian-Eulerian mesh (discrete Dirac function)
  - Very similar to actuator approach

H¢ l »¢ n− d¢ I w l ƒ − ˘ l B− ϱ¢
HP G



2. Immersed boundary method



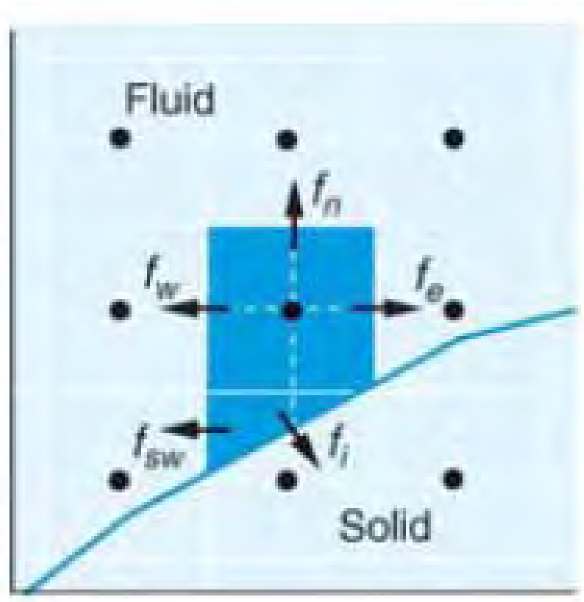
B. Sanderse, ‘An immersed boundary method for the flow over a circular cylinder’. Re=80

2. Immersed boundary method

- Discrete forcing
  - Ghost cell
    - Solution inside body
    - Simple implementation
    - Not strictly conserving mass, momentum, energy
  - Cut cell
    - Conservation properties satisfied
    - Much work involved for calculating geometric quantities
    - Small cell stability issues

2. Immersed boundary method

- Cut cell example



3. Vortex method

- Lagrangian formulation
- Particle method
- Blades with actuator-type model (?)

Pro:

+ Grid-free

+ No nonlinear convective term → **no artificial diffusion**

+ Larger time steps

+ Particles only needed in regions of non-zero vorticity

+ Wake meandering?

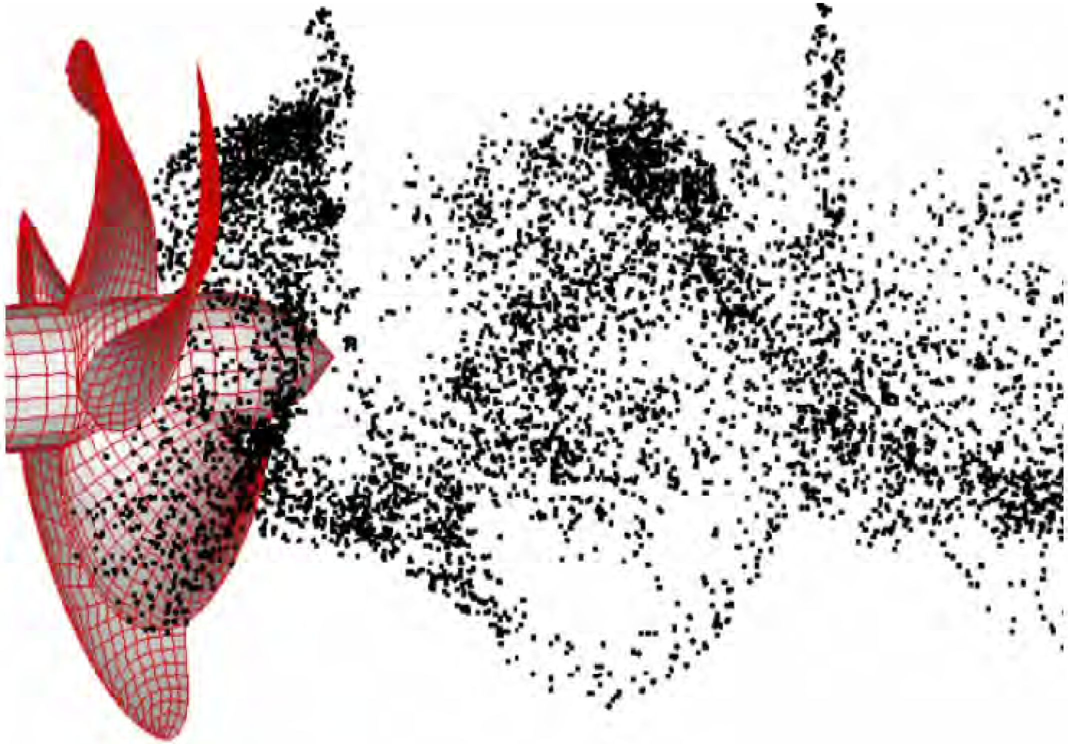
Con:

- Solution to N-body problem is costly (although fast algorithms exist nowadays)

- Particles needed in entire domain to represent ABL

- Handling no-slip boundaries

3. Vortex method



Zhao, Tsukamato: 'Hybrid vortex method for high Reynolds number flows around three-dimensional complex boundary', 2007.

Comparing methods - a test case

- Flow over a bluff-body, e.g. a cylinder or a fence: unsteady, viscous, vortex shedding
- Compare methods based on:

quality with which the wake is reproduced

depending on computational effort.

- Suggestions and feedback are welcome
- If interest, literature research can be made available on ECN site
- Thank you for your attention