CFD methods for wind turbine aerodynamics

Sørensen, Niels N.

Published in:
Extended abstracts

Publication date:
2008

Citation (APA):
CFD Methods For Wind Turbine Aerodynamics
21st Nordic Seminar on Computational Mechanics
October 16-17, 2008, Trondheim

Niels N. Sørensen
Wind Energy Department, Risø DTU
National Laboratory for Sustainable Energy
Risø’s history in brief

- **1956** Peaceful utilisation of nuclear energy
- **1976** Nuclear energy and other energy sources
- **1986** Energy research in general
- **1990** R&D with energy as the primary area
- **1994** State-owned enterprise
- **2000** The last nuclear reactor is decommissioned
- **2005** Impact within
  1. Technology for greater competitiveness
  2. Sustainable energy supply
  3. Health technology
- **2007** Merger with DTU, National Laboratory for Sustainable Energy
Outline of talk

• Aerodynamics of wind tubines
• CFD in Wind Turbine Aerodynamics
  – General Overview
  – The EllipSys2D/3D solvers
• Applications
  – Airfoil Aerodynamics
  – Transition modeling
  – Dynamic stall
  – Deep stall aerodynamics
  – Rotor aerodynamics
  – Aeroelasticity
  – Flow over terrain
Aerodynamics for wind turbines

- Flow over complex terrain
- Rotor aerodynamics
- Rotor/Tower interaction
- Wake aerodynamics
- Airfoil Flows
- Laminar/turbulent transition
- Hysteresis phenomena, dynamic stall
- Damping and stability
- Aeroelasticity
Development and origin of CFD for Wind Energy

• The application of numerical methods to fixed wing and rotor aerodynamics dates back to the late seventies in the aerospace community, solving steady Potential and Euler equations.

• The first unsteady solution to the Euler equations were seen through the eighties.

• With the continuous increase in computer power in the late eighties and early nineties the first Reynolds Averaged Navier-Stokes codes for helicopter applications appeared.

• With the possibility of handling viscous flow, the first applications of Navier-Stokes CFD solvers appeared in the wind turbine community in the late nineties.

• In Europe a series of EU-financed projects were providing the basis for many of these activities.
Components of a CFD method

The basic idea is to take the partial differential equations describing the fluid flow, transform them into a set of algebraic equations, and solve these using a numerical method on a computer.

Typical components of a CFD code are listed below:

- Mathematical Model
  - Compressible/Incompressible
  - Potential/Euler/Navier-Stokes
  - Turbulence Modeling
- Coordinate and basis vector systems
- Discretisation method, space and time
  - Finite Difference/Finite Volume/Finite Elements
- Solution Method
- Computational Grid
Turbulence Modeling

- Direct Numerical Simulation (DNS)

- Filtered Equations
  - Large Eddy Simulation (LES)

- Time Averaged Equations, Reynolds Averaged Navier-Stokes (RANS)
  - Algebraic Models (e.g. Baldwin-Lomax)
  - One Equations Models (e.g. Spalart-Allmaras, Baldwin-Barth)
  - Two Equation Models (e.g. k-ω, k-ε)
  - Reynolds Stress Models

- Hybrid Models
  - Detached Eddy Simulation
Laminar to Turbulent Transition

- Michel Model
- $e^n$-model (Data base)
- Bypass transition
- Correlation based transition
Computational Grid

- Structured
- Unstructured
- Multi-block
  - Different type of conforming grids
  - Overlapping or Chimera
- Hybrid Meshes (Structured/ Unstructured)
- Cartesian Cut Cells
Ellipsys2D/3D

The choices in our flow solver

- Incompressible Navier-Stokes
- Finite volume code (non-staggered)
- Cartesian or polar velocity components
- Patched multi-block grids (new overset option)
- Pressure/Velocity formulation
- Steady/Unsteady algorithm
- Moving Mesh/Moving Frame
- Turbulence Modelling RANS/DES
- Acceleration techniques: grid sequence/multi-grid
- Parallelized using MPI for distributed computers

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j) = 0 ,
\]

\[
\frac{\partial}{\partial t}(\rho U_i) + \frac{\partial}{\partial x_j}[\rho U_i U_j] - \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + \frac{\partial P}{\partial x_i} = S_v ,
\]
Airfoil Aerodynamics

2D applications

- Basic studies
- Cl/Cd/Cm for BEM computations
- Airfoil catalog
- Airfoil design and optimization
- Planning and conduction of measurements
- Laminar/turbulent transition
- Dynamic stall computations
Airfoil Aerodynamics
Laminar/Turbulent transition

NACA 64-618, Re=6.0 million
• C-mesh with 512x128 cells
• Wall normal distance: $y^+<1$
• Maximum expansion rate: $< 5\%$
• Inflow turbulence intensity 0.009\%
Airfoil Aerodynamics
Dynamic stall

- Dynamic stall
- Stall characteristics
- Aerodynamic damping
Deep Stall Aerodynamics
3D blade section in static stall

Unsteady lift time series

The development of a stochastic stall-model based on time series from DES-simulations is ongoing
Rotor Aerodynamics
NREL/Nasa Ames test

NASA Ames Tunnel (24.4x36.6 m)

NREL Phase-VI Wind Turbine
Rotor Aerodynamics

Blind Comparison

Upwind Configuration, Zero Yaw

Wind Speed (m/s)

Low-Speed Shaft Torque (Nm)

measurements
Risø comp.
Rotor Aerodynamics
NREL Phase-VI rotor, Cp. distribution

V=7m/s
Rotor Aerodynamics
NREL Phase-VI rotor, Cp. distribution

\[ V = 20 \text{m/s} \]
The transition model can improve the results in some cases.
Rotor Aerodynamics

NREL Phase-VI rotor, Laminar/turbulent transition

• Limiting streamlines
Rotor Aerodynamics

Yaw computations (60 degrees)

Azimuth variation of normal force
Rotor Aerodynamics
Rotors in atmospheric shear

- Rotors in shear flow have been studied using CFD to quantify hysteresis effects

\[ U(z) = 8 \text{m/s} \left( \frac{z}{90 \text{m}} \right)^{0.55} \]
Rotor Aerodynamics
Detailed design analysis

Enercon E-70 design

Tip-design
Rotor Aerodynamics
Drag values for parked wind turbine blades

Definition of aspect ratio
\( L/w = L^2/\text{Area} \)

<table>
<thead>
<tr>
<th>Blade</th>
<th>L/w</th>
<th>( C_d ) comp.</th>
</tr>
</thead>
<tbody>
<tr>
<td>LM8.2</td>
<td>11.4</td>
<td>1.23</td>
</tr>
<tr>
<td>LM19.1</td>
<td>18.0</td>
<td>1.32</td>
</tr>
<tr>
<td>Modern</td>
<td>21.0</td>
<td>1.16</td>
</tr>
</tbody>
</table>
**Rotor Aerodynamics**

**Rotor tower interaction**

Details of blade-tower interaction investigated in order to:
- study lock-in phenomena
- develop semi-emperical tower shadow model and noise model
Site Analysis
Complex Terrain

Terrains where WAsP is not suitable

Determining Speed-up, and flow inclination

Evaluation of turbine positions, from levels of turbulent kinetic energy
Conclusion

• A CFD methodology suited for wind energy has been introduced

• A series of applications of CFD to wind energy, including airfoil and rotor aerodynamics has been given, illustrating the possibilities.