Intelligent
Fluid
Solutions

Application of Modelling Methods in Wind Turbine Engineering

Andrej Horvat, David Hartwanger

Intelligent Fluid Solutions Ltd

Free flow turbines and their efficiency

May 06, 2011, University of Leicester
Leicester, UK

Content

- **Introduction**
- **Theoretical turbine prediction models**

  - Betz & actuated disk approach

  - Glauert & blade element method(s)
- **Computational Fluid Dynamics (CFD) approach**

  - 2D blade modelling

  - 3D blade and turbine modelling
- **Wind farm simulation and layout optimisation**
- **Summary**

Contact information

Dr. Andrej Horvat

Principal Engineer

Intelligent Fluid Solutions Ltd.

99 Milton Park, Abingdon, Oxfordshire, OX14 4RY, UK

Tel.: +44 (0)1235 84 15 05

Fax: +44 (0)1235 85 40 01

Mobile: +44 (0)79 72 17 27 00

Skype: a.horvat

E-mail: andrej.horvat@intelligentfluidsolutions.co.uk

Web: www.intelligentfluidsolutions.co.uk

IFS services

Idea-to-market process

How ?

Access to capital investment
Design evaluation and analysis
Patent application process assistance
Production methods and planning

Design development

How ?

Product technical assessment
Product performance assessment
Design optimisation
Full system modelling
In-house & outsourced exp. capabilities

Engineering analysis

How ?

World class engineering expertise
Continues R&D investment
State-of-the-art modelling tools
Access to electronic libraries

Areas of expertise and experience

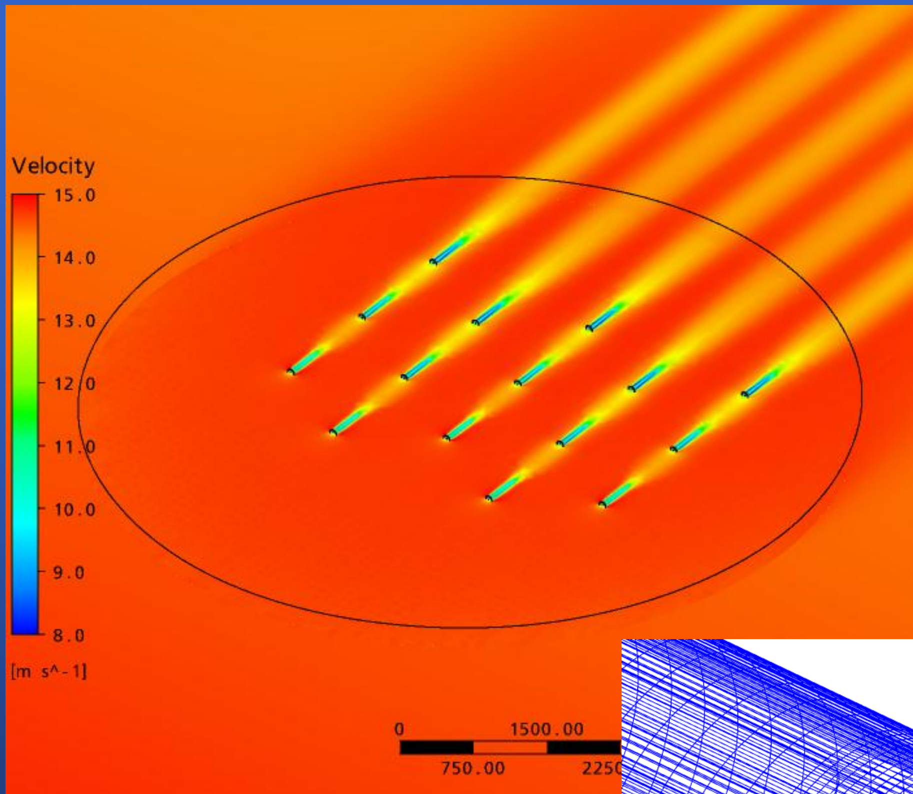
- Environmental Analysis
- Renewable Energy
- Nuclear Engineering
- Oil & Gas
- Combustion & Process Engineering
- HVAC & Fire Safety
- Transport
- General Industrial
- Consumer Products
- Sports Engineering
- Medical Engineering

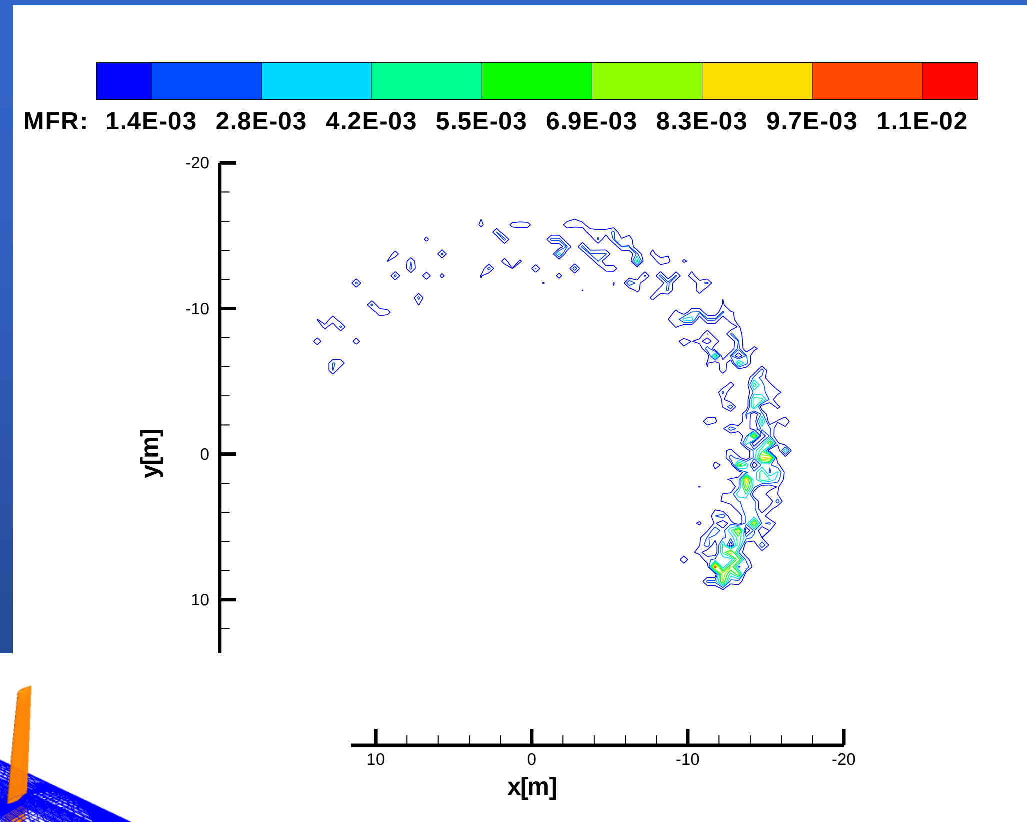
→ Environment

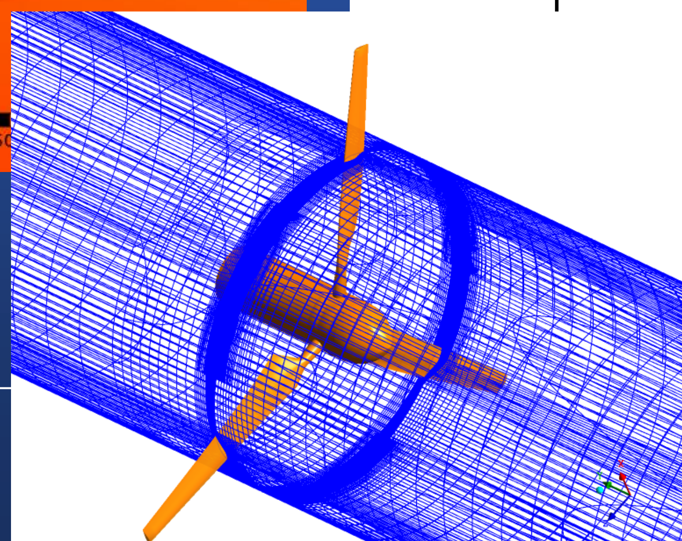
→ Energy

→ Process eng.

Areas of expertise and experience







Intelligent
Fluid
Solutions

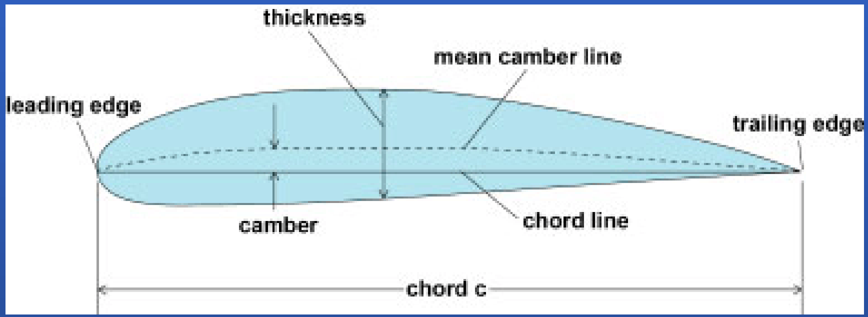
Introduction

Global energy demands and dwindling fossil fuels are making zero-head wind and water turbines increasingly important as renewable energy devices

- Device efficiency is a key parameter in the economic viability, recovery of initial capital investment and long term profitability of such devices.
- To determine a performance curve of an operational turbine, a range of aspects need to be taken into account. These include complex fluid dynamics, local topology and other environmental conditions, structural loadings and vibrations, transmission loads and generator dynamics.
- Furthermore, in most cases, turbines are installed in groups or farms. Flow interaction in a multiple turbine installation reduces the power output and thus has an important influence on economics of whole installations.

Introduction

Design of the modern wind turbine rely on the same aerodynamic principles as an aircraft



Wind turbines fall into two main categories, those that depend upon aerodynamic drag to drive them (i.e. old style windmills) and those that depend upon aerodynamic lift.

Intelligent
Fluid
Solutions

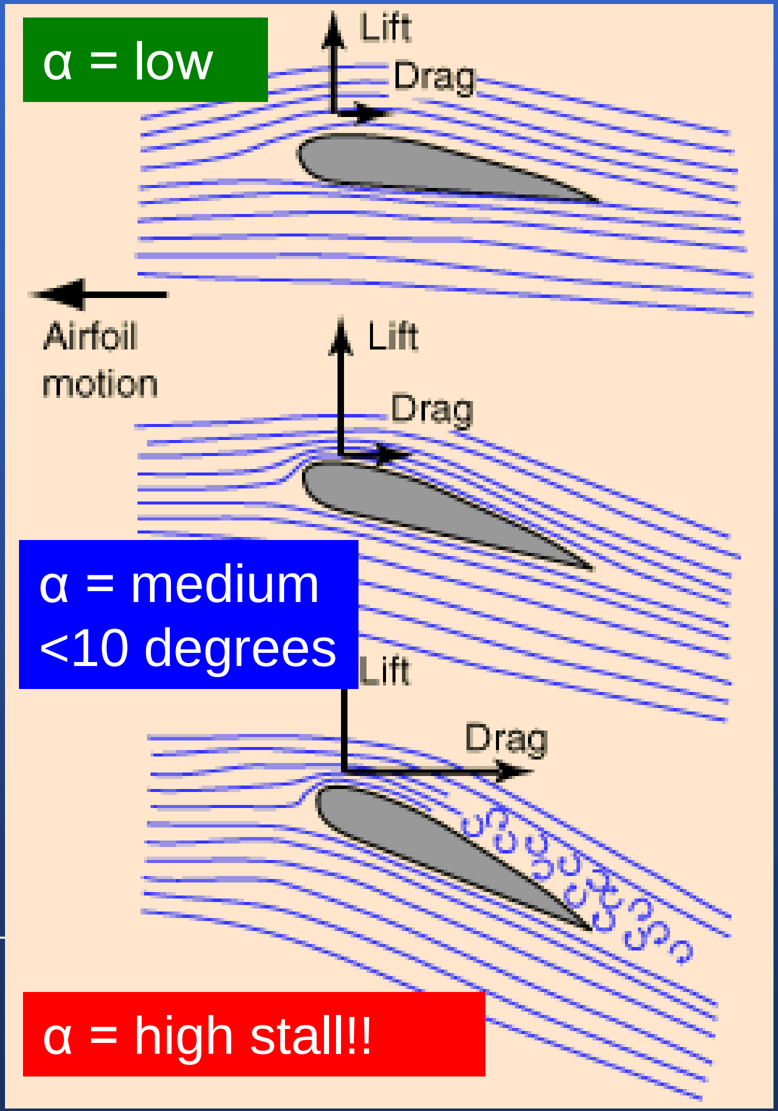
Introduction

The lift and the drag forces strongly depend on the incidence angle

- Lift Force is perpendicular to the direction of motion

- Drag Force is parallel to the direction of motion

Modern wind turbine design - the lift force is maximized, whereas the developed drag force stays small (1 to 2% of the lift force in pre-stall conditions)

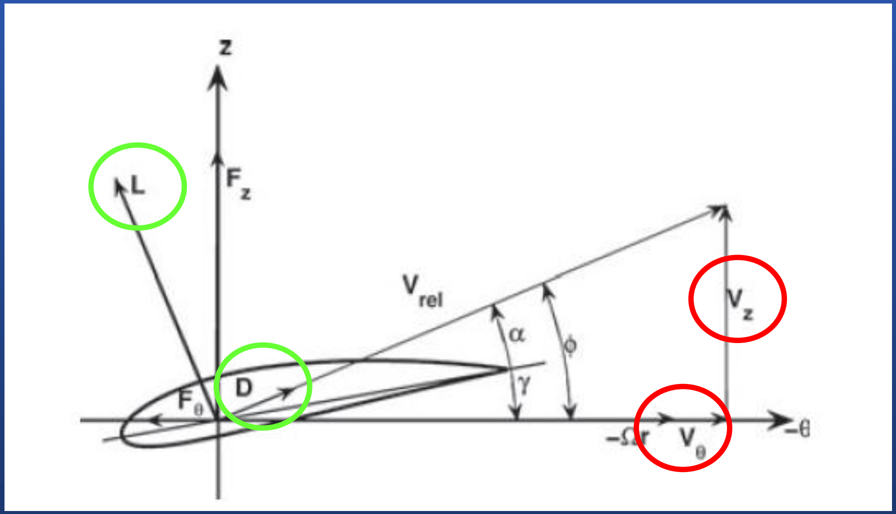


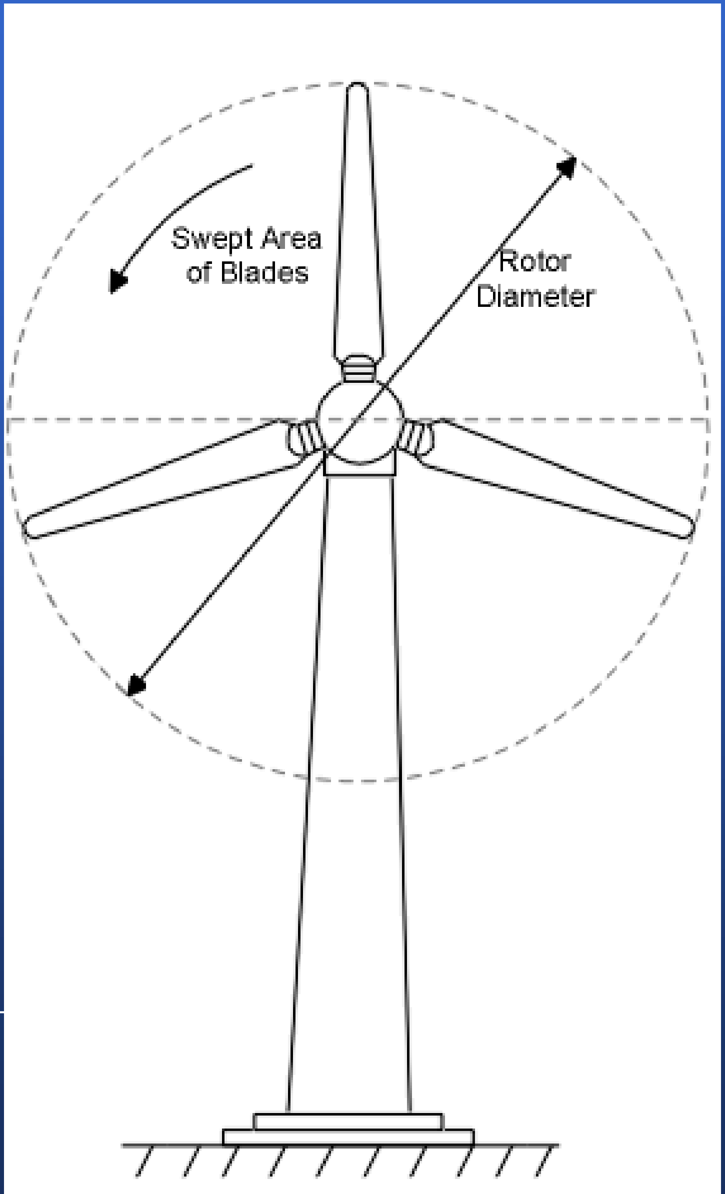
Intelligent
Fluid
Solutions

Introduction

The turbine rotates with angular velocity Ω and tangential velocity Ωr

Wind with local velocity components v_z and v_θ creates lift force L and drag force D, which at a correct incidence angle α turn around the turbine rotor





Intelligent
Fluid
Solutions

courtesy of ESN

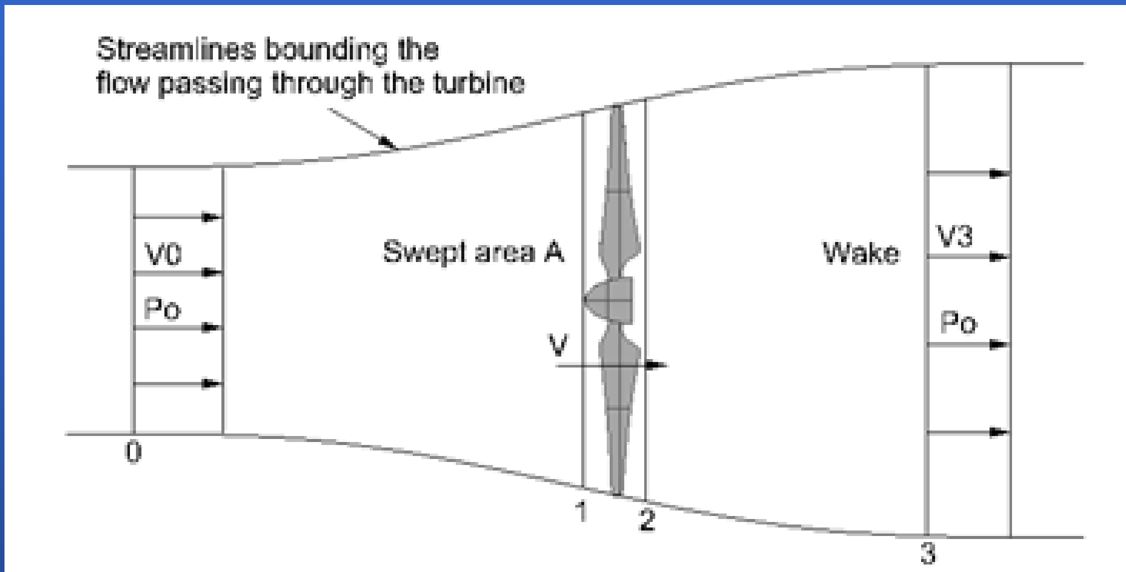
Theoretical turbine prediction models

The simplest model (Betz, 1926) that can be used to predict wind turbine power output (and, subsequently, turbine efficiency) represents the rotor as an actuator disc, which creates a pressure discontinuity i.e. acts as a momentum sink

The Actuator Disc approach uses the following assumptions:

- The flow is ideal and rectilinear across the turbine i.e. steady, homogenous, inviscid, irrotational, and incompressible
- Both the flow and thrust are uniform across the disk
- The static pressure at the upwind and downwind boundaries is equal to the ambient static pressure

Theoretical turbine prediction models



$$P = F_a v$$

$$P = \dot{m}(v_0 - v_3)v = (p_2 - p_1)A$$

Energy loss by the wind is

$$P = \dot{m}(v_0^2 - v_3^2)/2$$

and therefore $v = (v_0 + v_3)/2$

- The wind exerts an axial thrust on the turbine in the flow direction
- An equal and opposite force is exerted by the turbine on the wind through its mounting with the ground

Theoretical turbine prediction models

More often the following form is derived

$$P = 2\dot{m}(v_0 - v)v = 2\ \ Av^2(v_0^2 - v^2)$$

The fractional decrease in wind velocity between the free stream and rotor plane can be expressed in terms of an axial induction factor a:

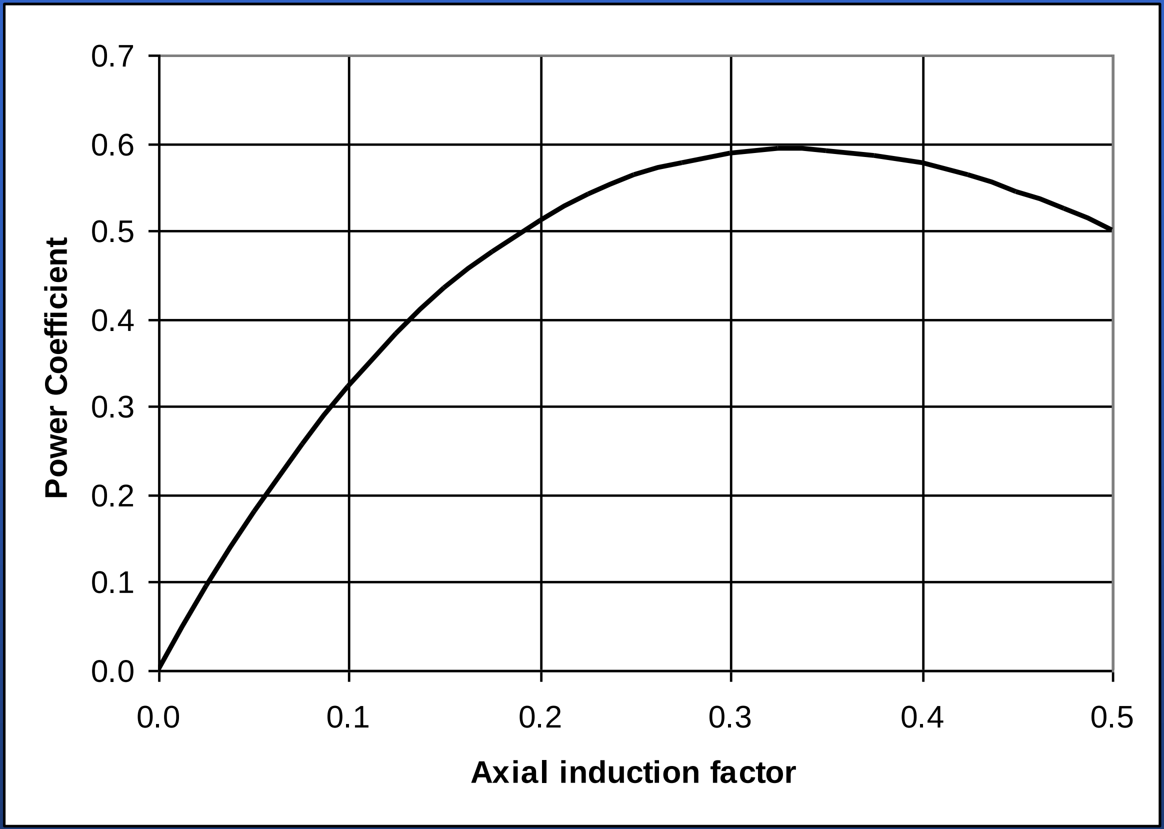
$$a = (v_0 - v)/v_0$$

Then the expression for turbine power output is $P = 2\ \ Av^3 a(1-a)^2$

and the power coefficient (i.e. the extracted power over the total available power) is simply

$$C_P = \frac{P}{\frac{1}{2}\ \dot{m}\ v_0^2} = 4a(1-a)^2$$

Theoretical turbine prediction models



The maximum value of $a = 1/2$; requires $v_3 \rightarrow 0$

C_p reaches maximum of 0.593 when $a = 1/3$

The corresponding downstream wake velocity, $v_3 = v_0 / 3$ and the wake area A_3 is double the turbine swept area A

This is known as the Betz limit for an ideal frictionless turbine

Theoretical turbine prediction models

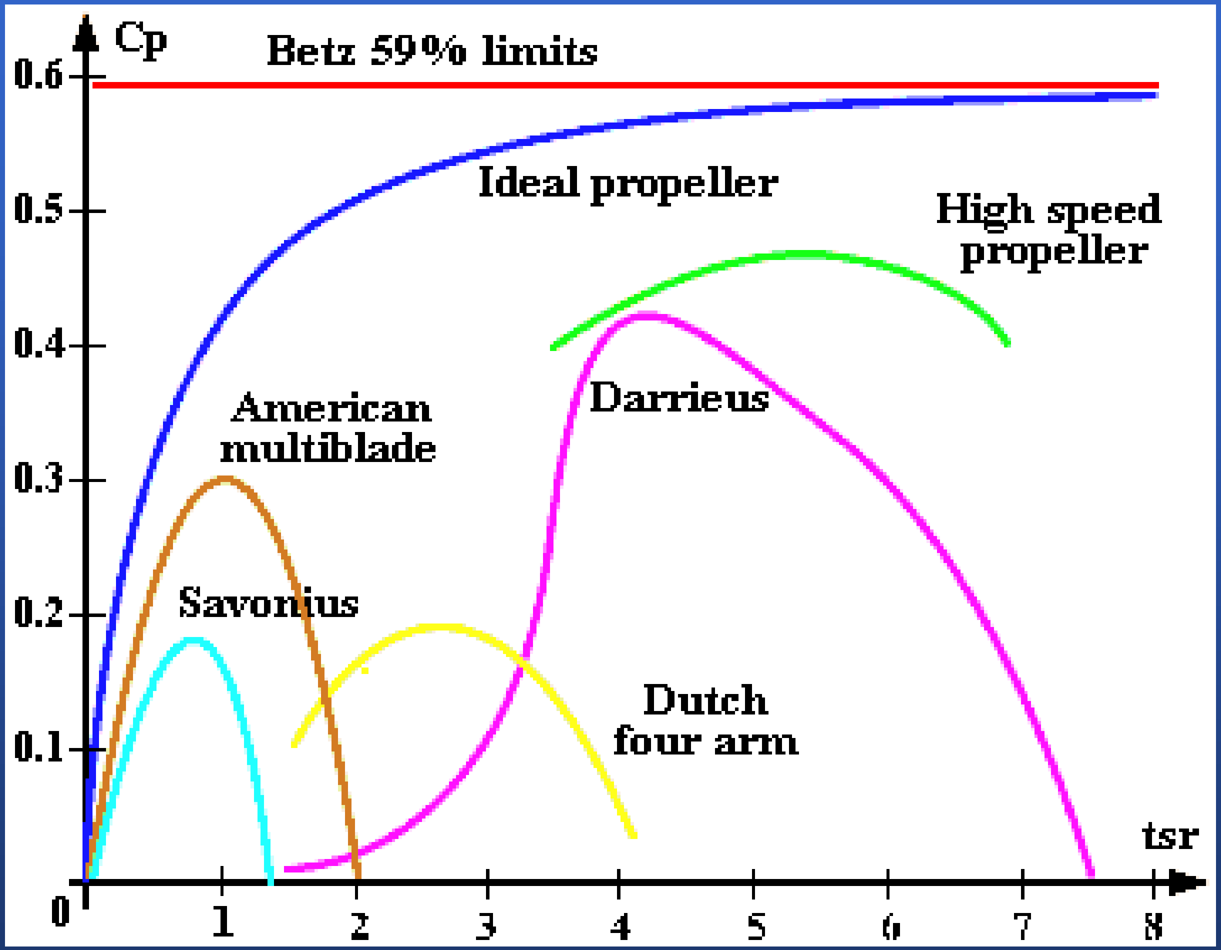
All wind power cannot be captured by rotor or air would be completely still behind rotor

Theoretical limit of rotor efficiency is 59%

In reality a real wind turbine does not achieve this efficiency level due to:

- Rotation in the wake caused by the reaction with the spinning rotor
- A non-uniform pressure distribution in the turbine plane
- Aerodynamic drag due to viscous effects
- Energy loss due to vortices at the blade tips

Theoretical turbine prediction models



Theoretical turbine prediction models

The impact of the rotating wake can be estimated by extending the Betz analysis to a 2-D model in the radial direction (Glauert, 1935)

The control volume is divided into many non-interacting elementary radial blade sections that could be analysed independently (Blade Element Method)

- The flow far upstream is purely axial; however there is a discontinuous jump in angular velocity across the rotor plane because torque is exerted on the rotor
- The turbine wake rotates in the opposite direction to the rotor with an angular velocity ω_2
- Circumferentially averaged torque on a radial element is then

$$d\tau = \omega_2 r^2 d\dot{m} = \omega_2 r^2 (\rho v_2 2\pi r dr)$$

Theoretical turbine prediction models

- Blade element model usually utilizes an angular induction factor and a local speed ratio:

$$a' = \frac{\omega_2}{2\Omega} \qquad \lambda_r = \frac{\Omega r}{v_0}$$

- The turbine torque and power are calculated by integrating radial contributions

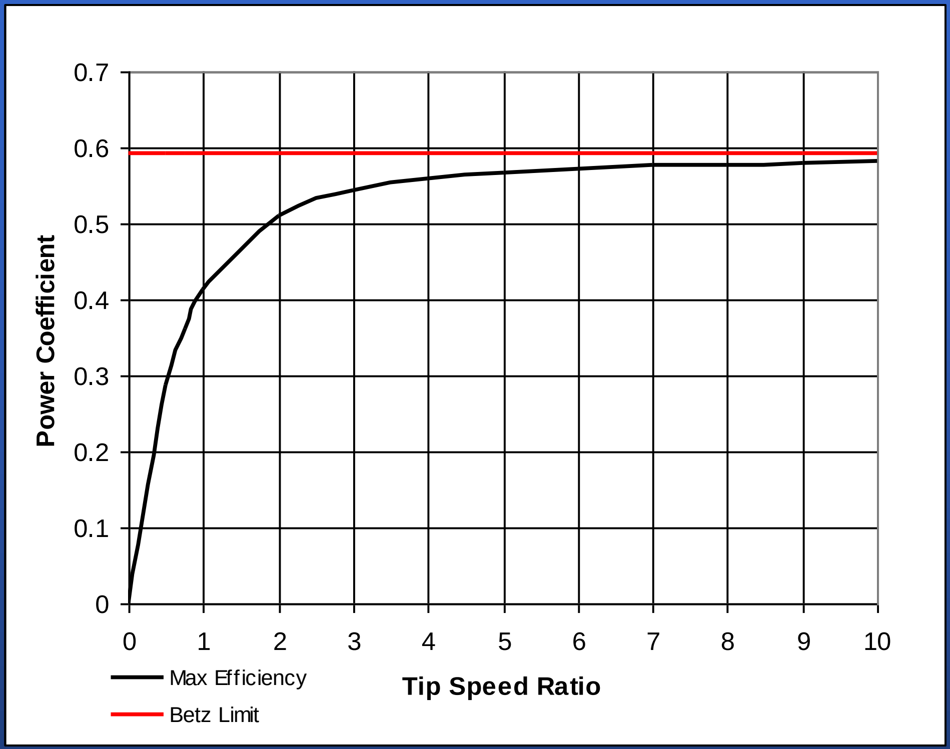
$$\tau = 4\pi\rho\Omega v_0 \int_0^R (1-a)a' r^3 dr \qquad P = \tau\Omega$$

where a and a' are radial functions, which need to be empirically determined

- Also, it can be shown for an ideal turbine the the axial and angular induction factors are related

$$a' = \frac{1}{2}\left(\sqrt{1 + \frac{4a}{\lambda_r^2}(1-a)} - 1\right)$$

Theoretical turbine prediction models



Power coefficient C_p when wake rotation is taken into consideration

Turbines with high tip-speed ratio (TSR) and low angular induction factor (a') are able to deliver more power

Theoretical turbine prediction models

- Glauert’s wake rotation model is still subject to the assumptions of a uniform distribution in tangential direction and zero radial velocity in the turbine plane
- Due to these assumptions, the model overestimates forces and torque applied to the turbine
- Therefore, Glauert’s model asymptotes towards the Betz limit as the tip-speed ratio tends to infinity, i.e. as the rotation in the wake tends to zero

More recently, in 2001, Gorban, Gorlov and Silantyev introduced an alternative model, that considers non-uniform pressure distribution and curvilinear flow across the turbine plane

Theoretical turbine prediction models

One of the variations of the approach is the Actuator Line Model:

- The blade relative velocity v_{rel} and incidence angle α are calculated from the local wind velocity components

$$v_{rel} = \sqrt{v_z^2 + (\Omega r - v_\theta)^2} \qquad \alpha = \varphi - \gamma$$

  where relative flow angle is $\varphi = \tan^{-1}\left(\frac{v_z}{\Omega r - v_\theta}\right)$ and γ is local blade pitch angle

- Based on local values of Reynolds number (Re) and the incidence angle (α), lift (C_L) and drag (C_D) coefficient are selected from a predefined database

- Lift and drag force per spanwise length of the blade are then expressed as

$$F_L = \frac{1}{2} C_L \rho |v_{rel}| v_{rel} l \qquad F_D = \frac{1}{2} C_D \rho |v_{rel}| v_{rel} l$$

Theoretical turbine prediction models

- Axial and tangential force acting on a blade segment are defined as

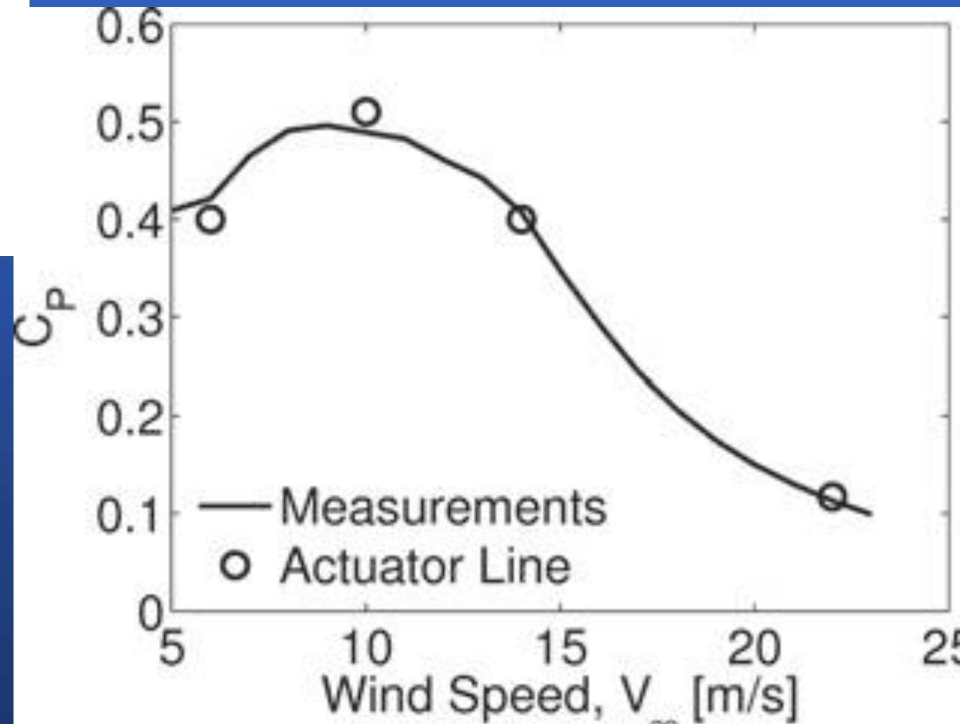
$$F_z = F_L cos(\varphi) + F_D sin(\varphi) \qquad F_\theta = F_L sin(\varphi) - F_D cos(\varphi)$$

- The torque produce by each blade is then obtain by integration in the radial direction

$$\tau = \int_0^R r F_\theta dr$$

Theoretical turbine prediction models

N. Troldborg et al.
(Wind Energ., 2009)



Wake visualisation using vorticity isosurface

Power coefficient comparison for the Tjareborg wind turbine

Intelligent
Fluid
Solutions

Computational Fluid Dynamics (CFD) approach

- Solves discretized and averaged Navier-Stokes equations for the flow around turbine blades
- Due to turbulence motion, the transport equations are averaged over time (RANS turbulence models) or filtered using a spatial filter (LES turbulence models)

$$\partial_t\bar{\rho} + \partial_j\left(\bar{\rho}\widetilde{v}_j\right) = 0$$

$$\partial_t\left(\bar{\rho}\widetilde{v}_j\right) + \partial_i\left(\bar{\rho}\widetilde{v}_i\widetilde{v}_j\right) = -\partial_j\bar{p} + 2\partial_i\left(\mu\overline{S_{ij}}\right) + \bar{\rho}g_j - \partial_i\left(\bar{\rho}\Pi_{ij}\right)$$

$$\bar{\rho}\Pi_{ji} = \overline{\rho v_j v_i} - \bar{\rho}\tilde{v}_j\tilde{v}_i \Rightarrow \overline{\rho v_j^* v_i^*}$$

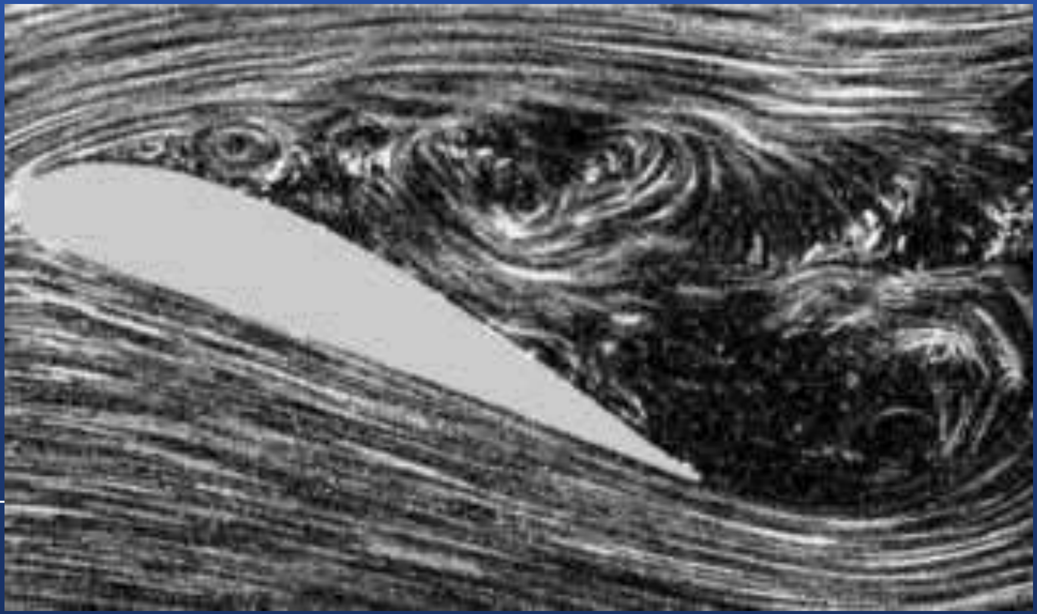
$$\bar{\rho}\Pi_{ij} - \frac{1}{3}\bar{\rho}\Pi_{ll}\delta_{ji} = -2\mu_t S_{ij} + \frac{2}{3}\mu_t(\partial_l\tilde{v}_l)\delta_{ji}$$

Turbulent stresses, in most cases modelled using the eddy viscosity concept

Computational Fluid Dynamics (CFD) approach

CFD can provide three-dimensional and time variation of all flow parameters, nevertheless, accuracy of the numerical prediction crucially depends on

- Resolution of the numerical grid i.e. number of computational points and their distribution
- Suitability of turbulence model to capture complex flow behaviour i.e. boundary layer transition, separation, transient effects, trailing and tip vortices



Jaganath (GNU)

Intelligent
Fluid
Solutions

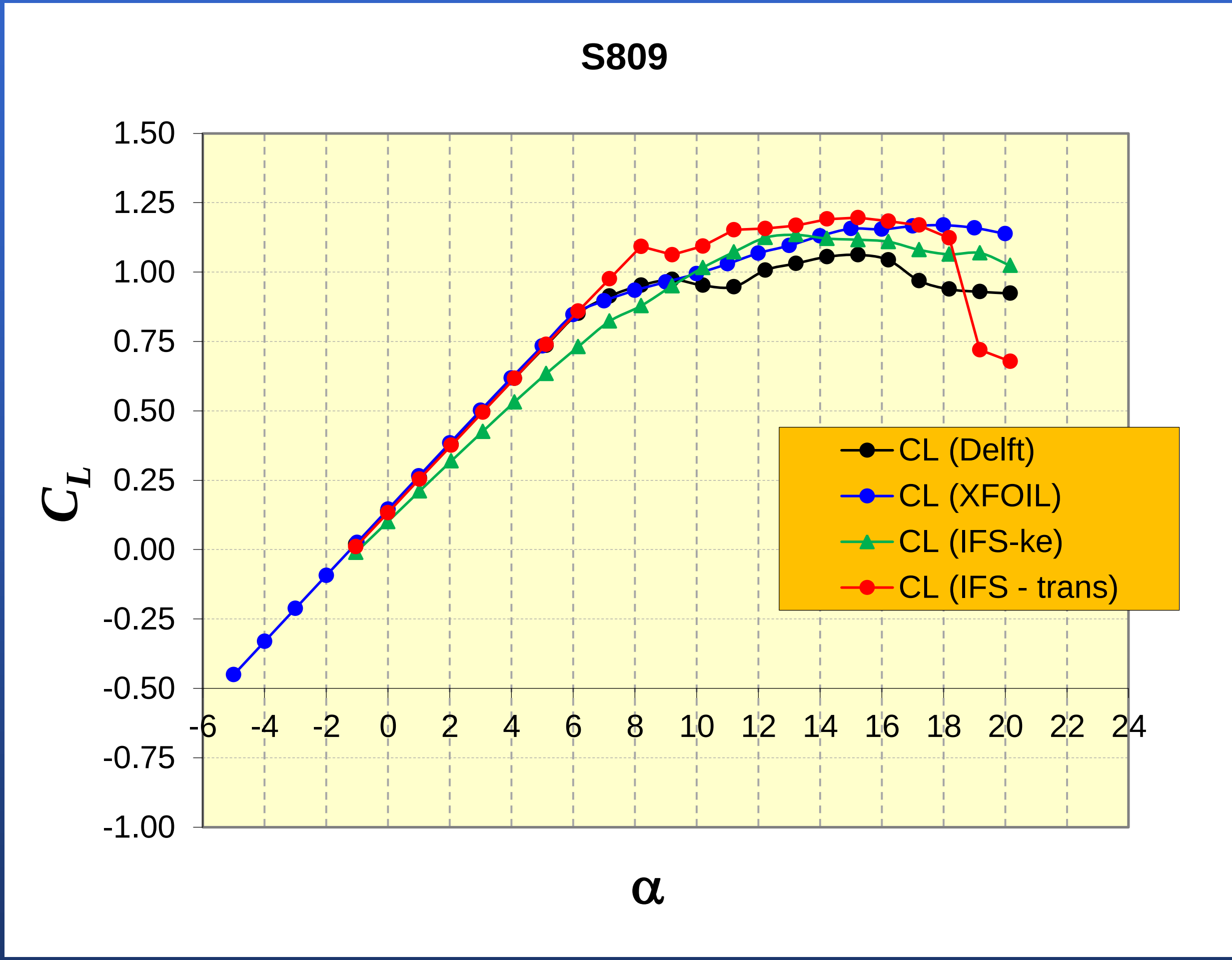
Computational Fluid Dynamics (CFD) approach

The required computational resources for CFD simulations of a turbine during its operation are extremely large, therefore, different simplifications and empirical models are introduced or coupled with the CFD model:

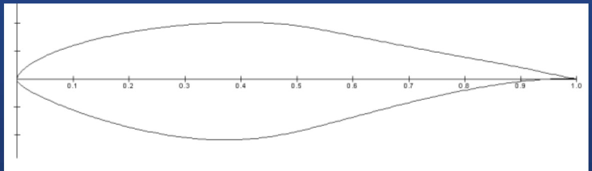
- Two dimensional CFD simulations of a wing section may be utilized to calculate drag and lift force (or drag and lift coefficient) for different incidence angles

- Such database $C_L(Re, \alpha)$ and $C_D(Re, \alpha)$ can then be used to integrate forces along the blade and to calculate torque and the turbine power output

Computational Fluid Dynamics (CFD) approach

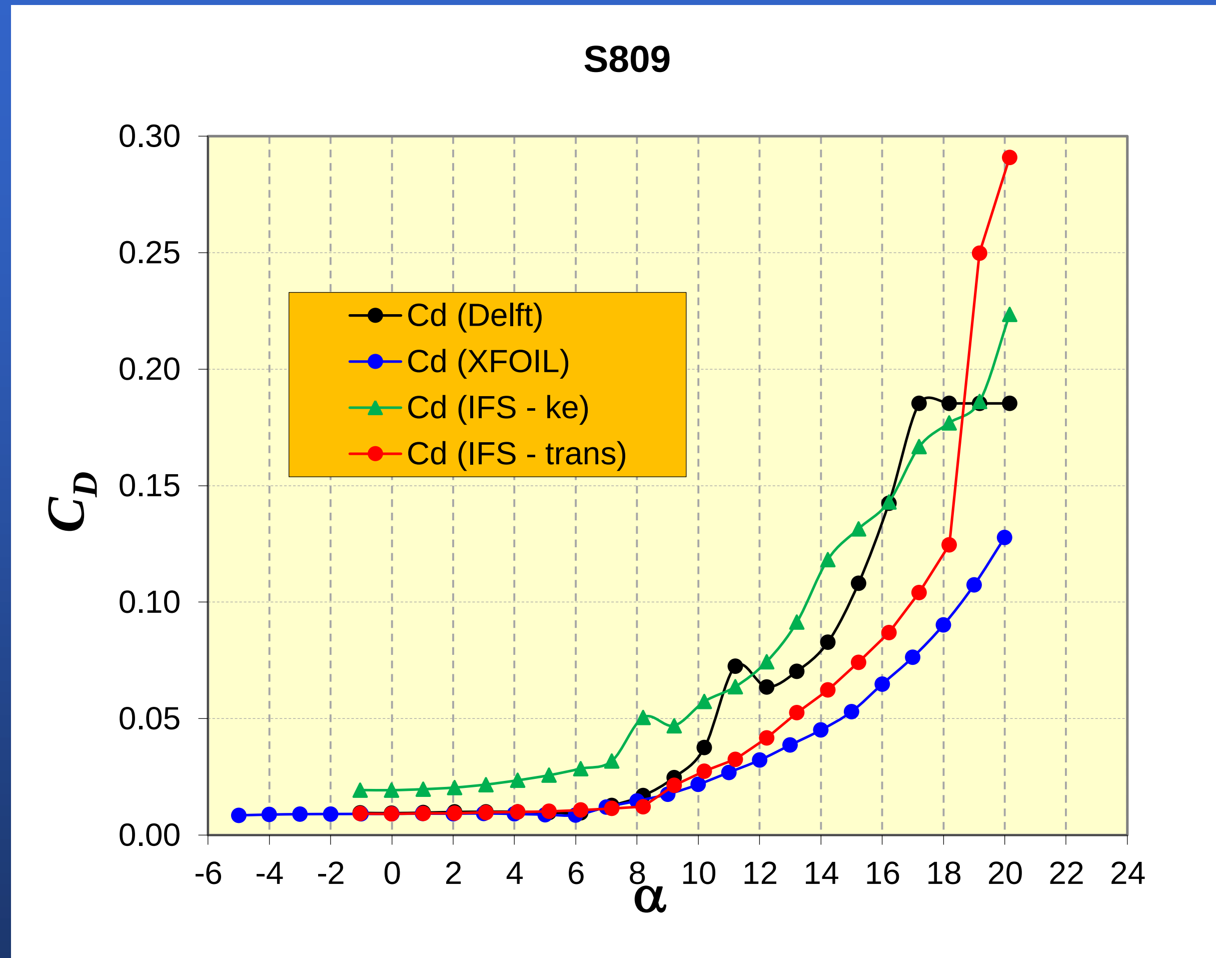


Lift coefficient comparison for airfoil S809

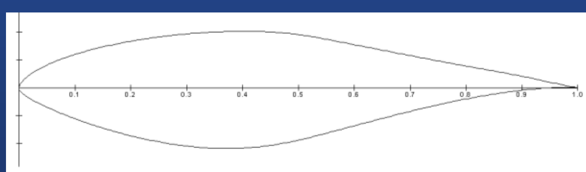


Intelligent Fluid Solutions

Computational Fluid Dynamics (CFD) approach



Drag coefficient comparison
for airfoil S809



Computational Fluid Dynamics (CFD) approach

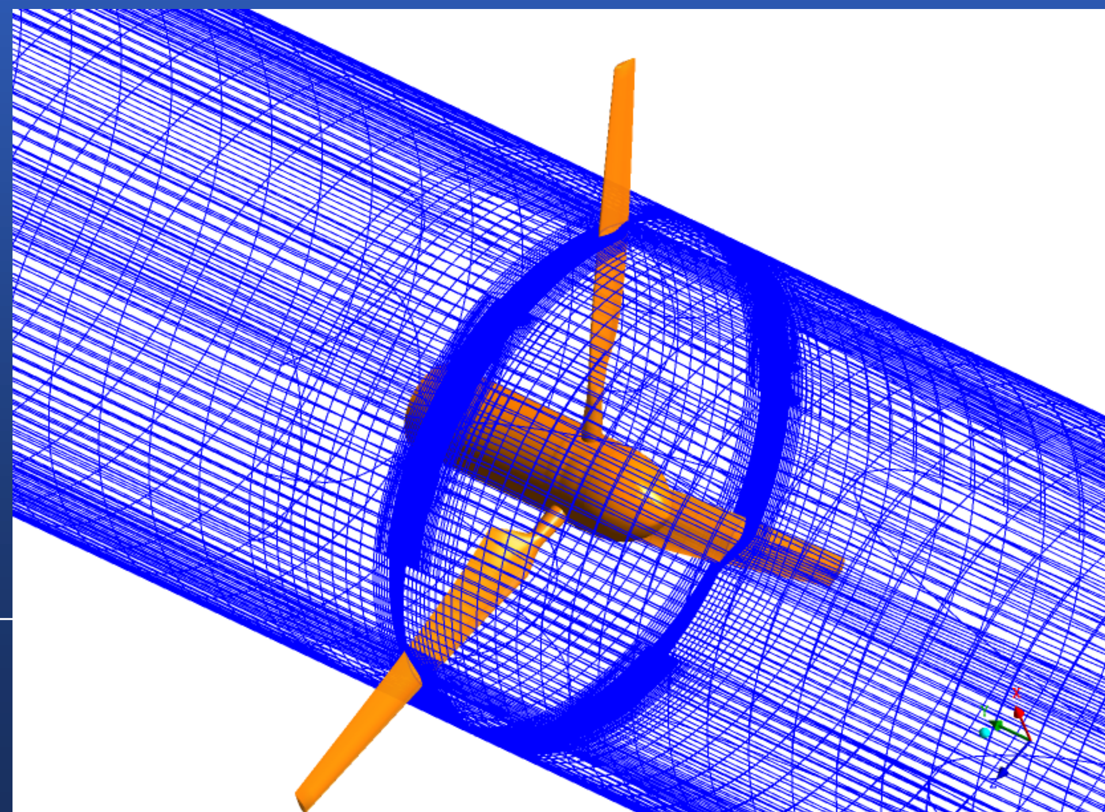
The 2D CFD simulations of turbine segments cannot take into account

- Radial flow effects
- Influence of the nacelle and the support tower on the flow behaviour
- Three-dimensional effects of the wake formation
- Surrounding topology
- Environmental effects – ground surface boundary layer formation, buoyancy, flow stability etc

Computational Fluid Dynamics (CFD) approach

Three dimensional CFD simulations do overcome deficiencies of 2D model, although they are rarely performed for engineering purposes:

- require numerical grids of at least few million of nodes
- due to transient flow effects, they need to be run transiently with significantly small timestep



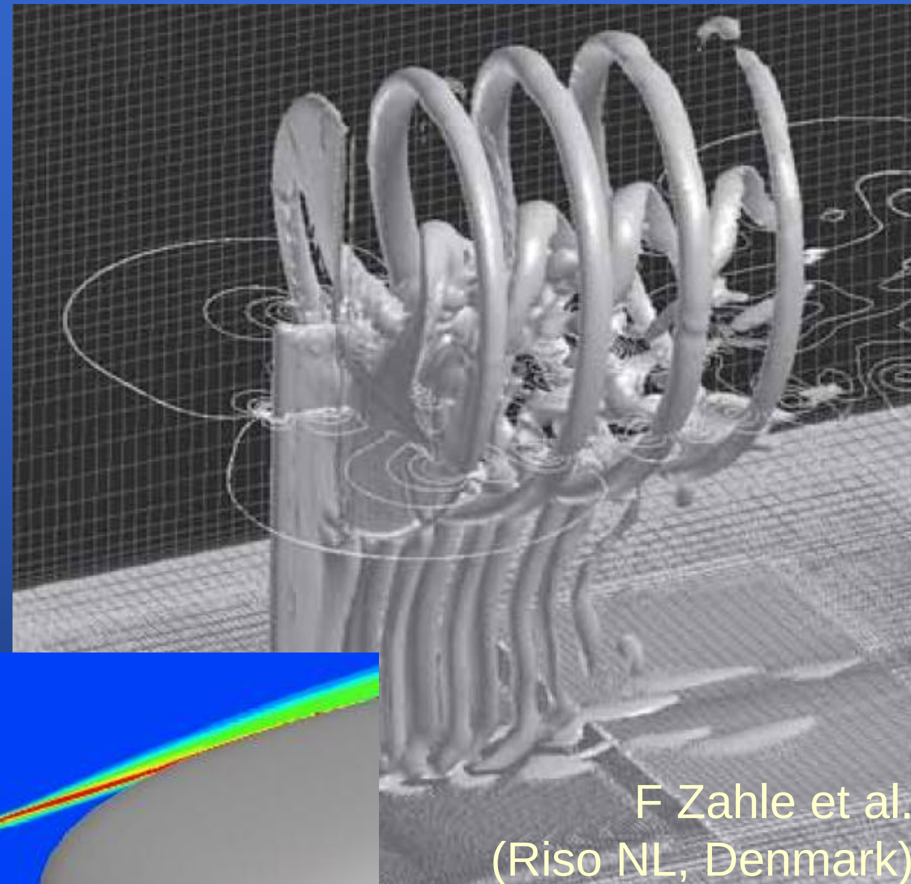
Section of the numerical mesh for UAE turbine

Intelligent
Fluid
Solutions

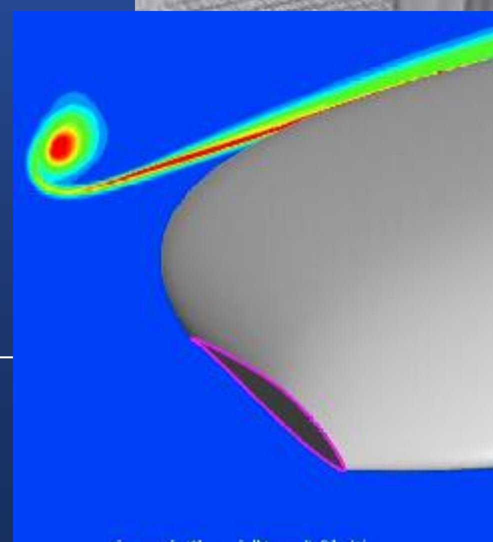
Computational Fluid Dynamics (CFD) approach

Three dimensional CFD simulations allow investigation of

- Blade design details
  (radial profile and pitch changes)
- Influence of nacelle and support tower
- Difficult flow physics conditions
  (tip vortex, stall conditions etc)

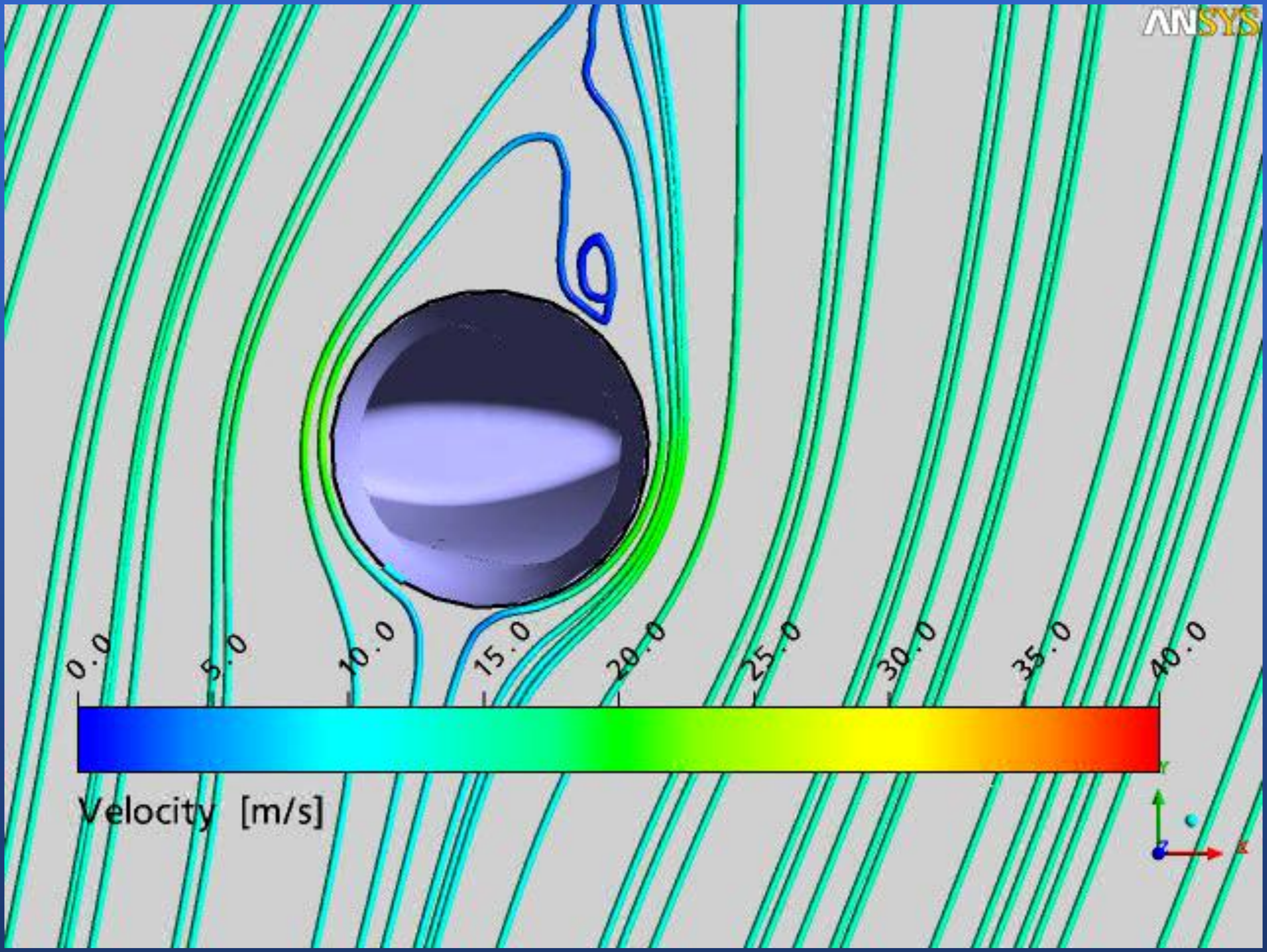


F Zahle et al.
(Riso NL, Denmark)



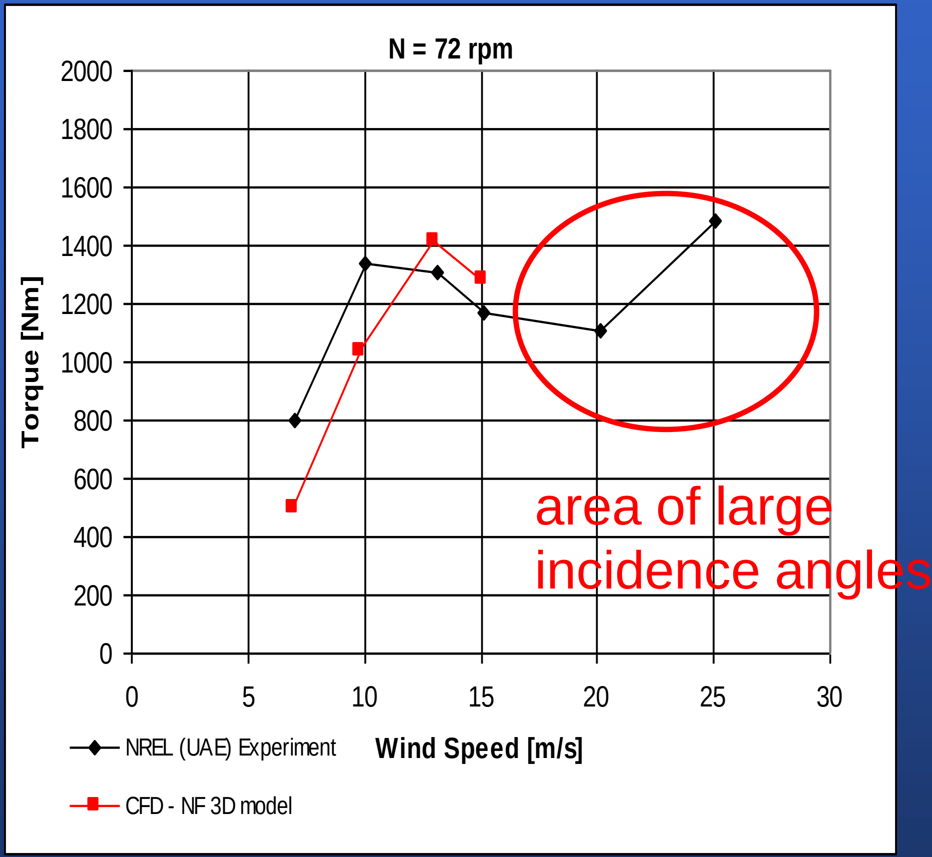
S. Herr et al.
(GE Wind Energy)

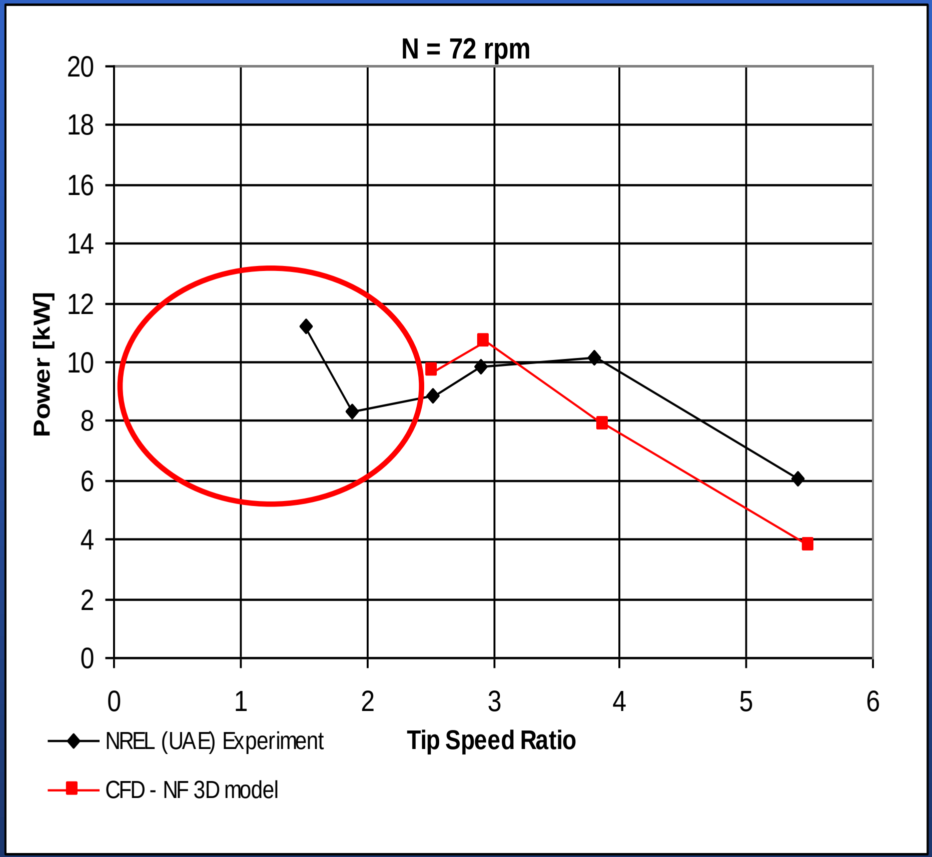
Computational Fluid Dynamics (CFD) approach



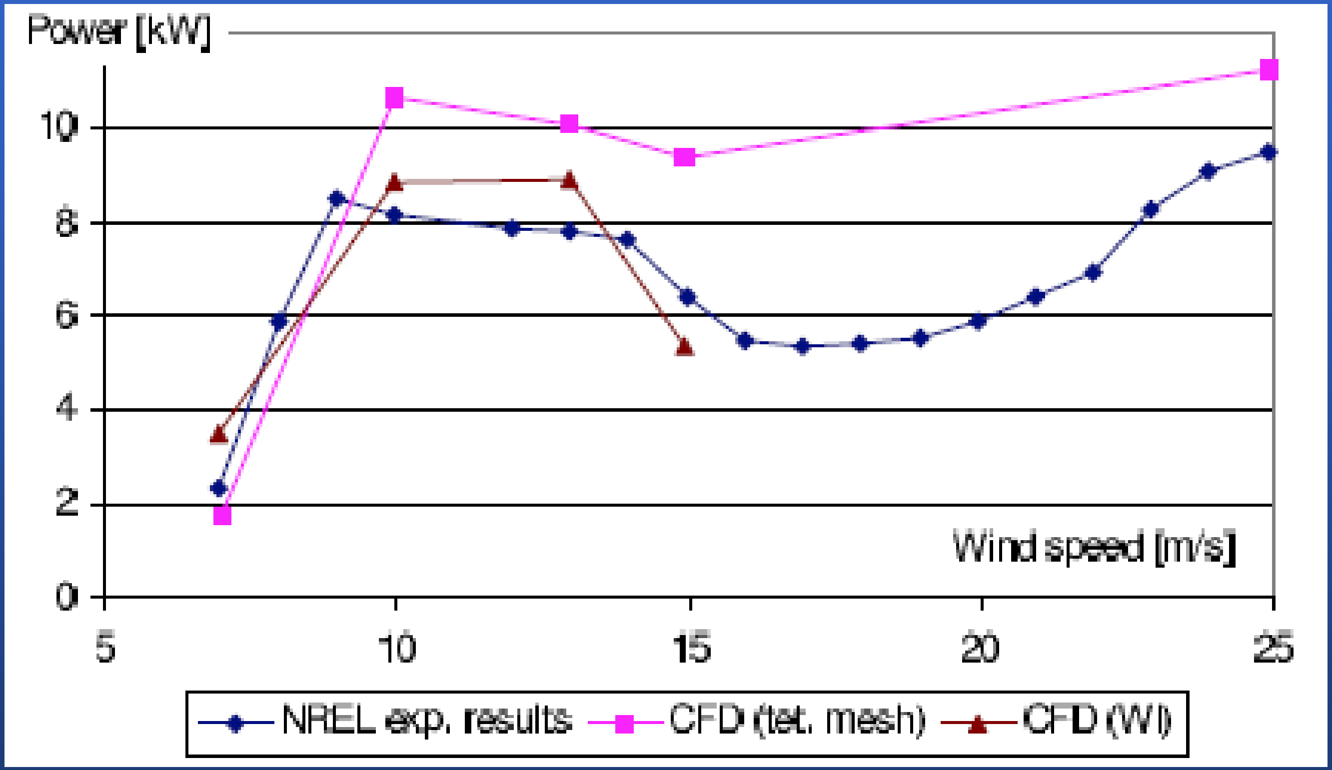
Computational Fluid Dynamics (CFD) approach

How accurate are flow predictions using 3D CFD modelling ?





Computational Fluid Dynamics (CFD) approach



S. Herr et al.
(GE Wind Energy)

Wind farm modelling and optimization

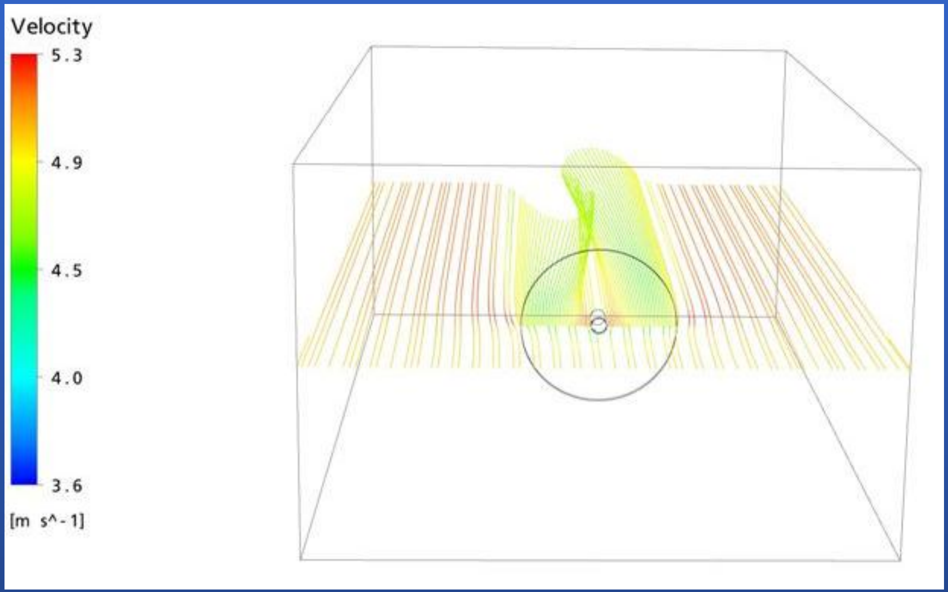
Accurate prediction of wind farm power output is necessary for planning installation capacity and to maximise return on the investment

Beside flow physics associated with individual turbines, modelling need to include

- wake interaction
- topology
- environmental conditions (i.e. ground boundary layer, buoyancy effects)

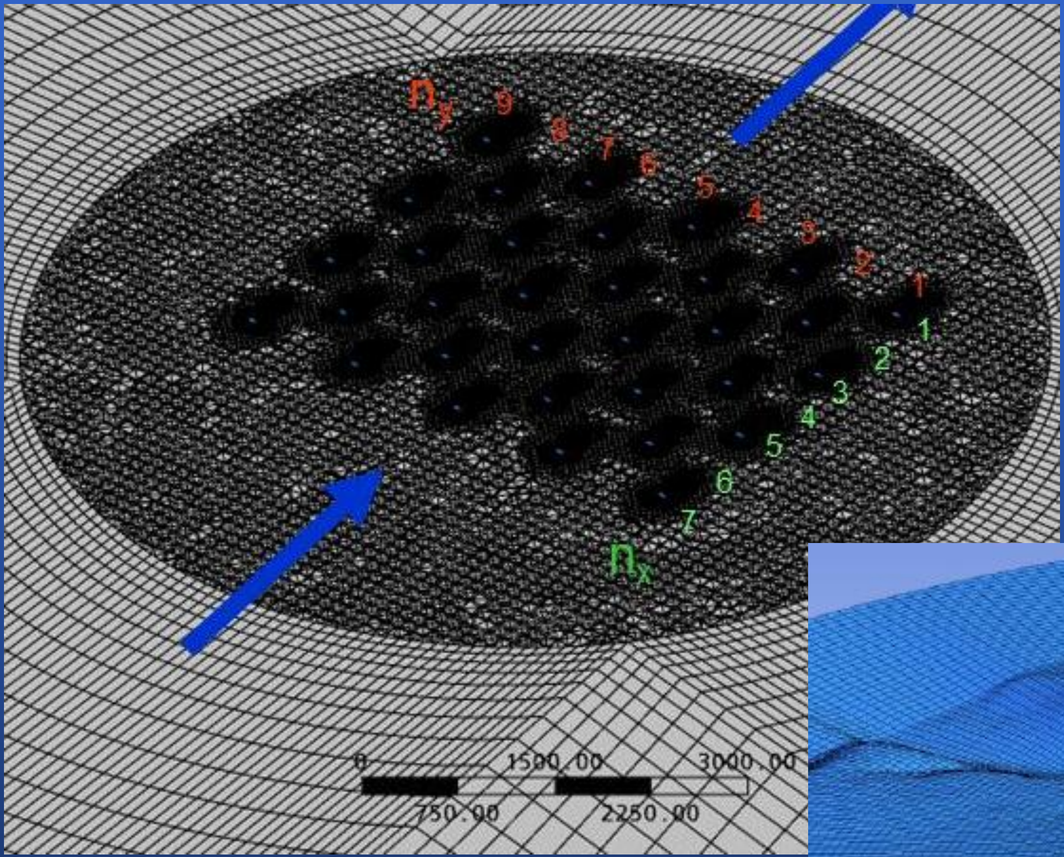
Combination of CFD approach and actuator disc (or blade element) model is necessary to reduce computational requirements

Wind farm modelling and optimization

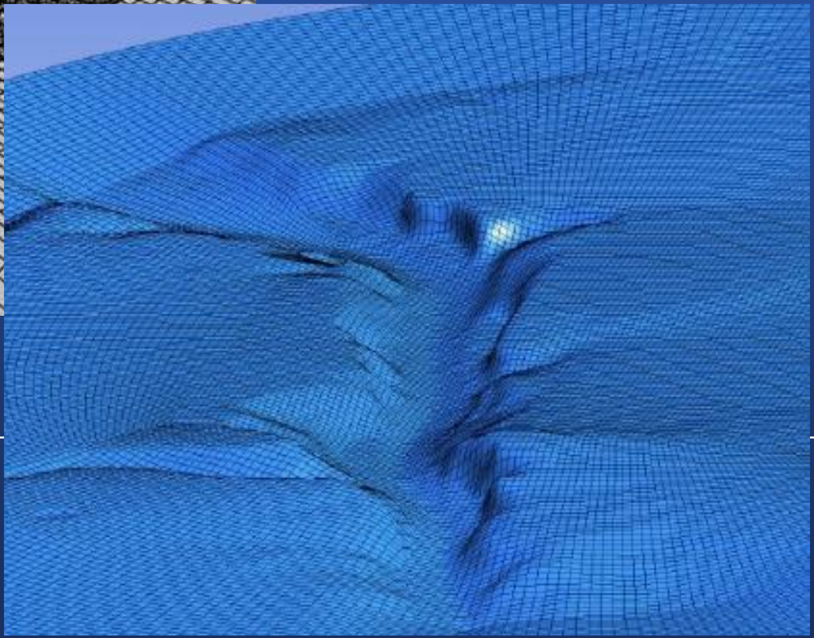


- Based on local velocity distribution, actuator disc (or blade element) model calculates forces and power output
- Effects of a rotating turbine on the flow are modelled with momentum sources/sinks
- Correct time-averaged representation of axial and tangential wake velocities

Wind farm modelling and optimization

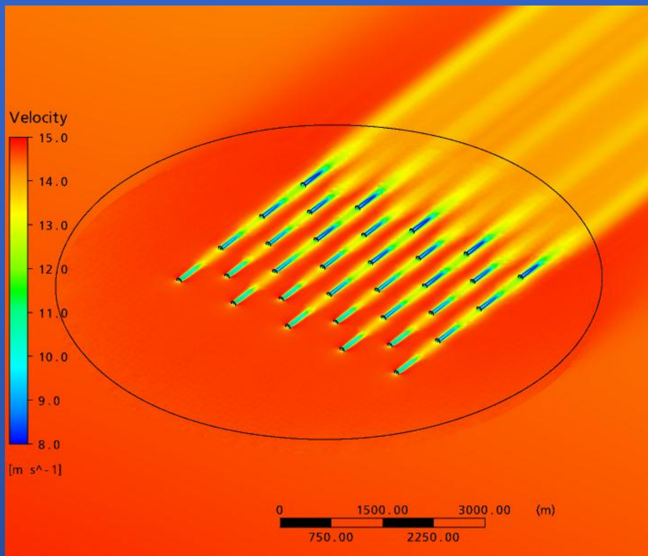


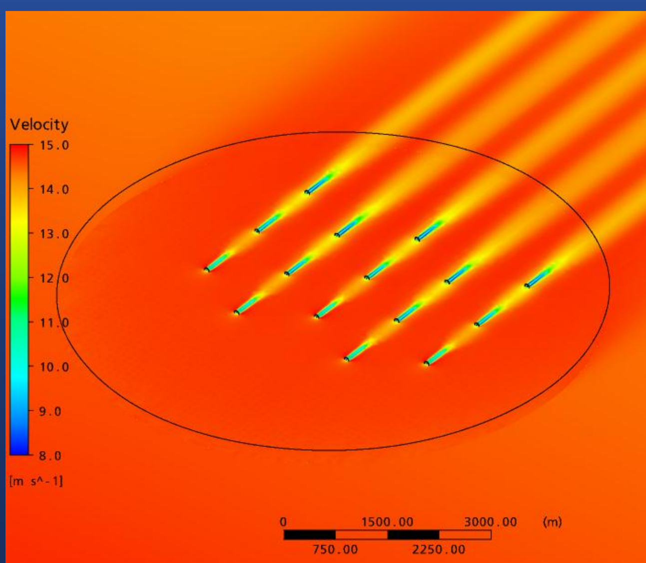
- The turbine (actuator disc) model is then inserted in a CFD model of a wind farm
- Different turbine arrangements can be analysed to minimize wake effect
- Real topology environment can be assessed



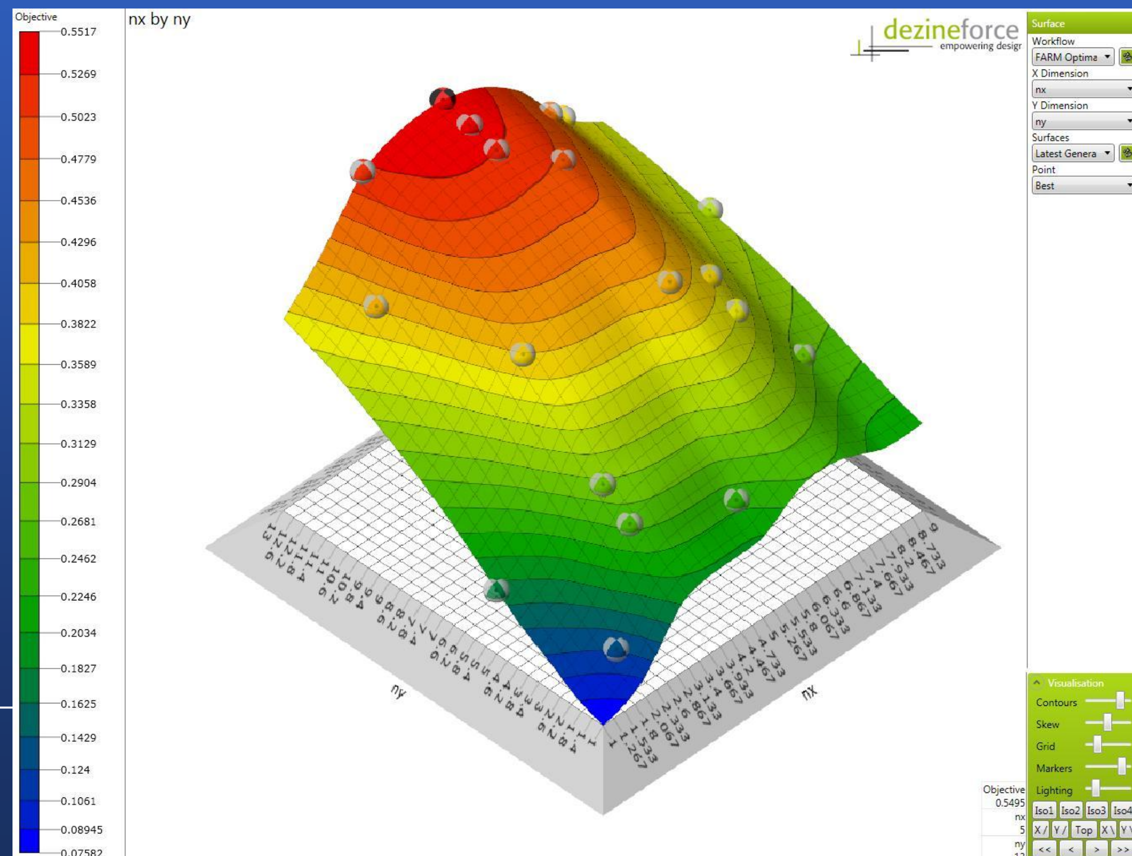
(ANSYS)

Wind farm modelling and optimization





- Such combine model can be coupled with an optimizer to maximize power output for a given investment



Summary

- The design of modern wind turbines rely on the same aerodynamic principles as an aircraft - predicting lift and drag force in a challenging environmental conditions
- Actuator Disc (Betz) approach gives an approximation of the turbine maximum power output
- Blade Element (Glauert) model introduces radial variation of the flow condition and wake rotation, where power output is related to the turbine tip-speed ratio
- Although useful as a first approximation, both models overestimate the power output
- Actuator Line model relies on databases of the lift and drag coefficient as functions of local flow conditions
- The accuracy power prediction depends on the accuracy of utilized databases

Summary

- Using Computational Fluid Dynamics (CFD) modelling, lift and drag forces (or lift and drag coefficient) are calculated from local flow and geometrical conditions
- Flow physics is captured much more accurately, which considerably improves accuracy of the turbine power output prediction
- Despite advance turbulence modelling, prediction of stall conditions is still (at least) challenging
- Especially for 3D CFD simulations, large computational resources are needed which limits their use in engineering practice
- For wind farm analysis and optimization of their power output, combine models (e.g. CFD & actuator disc model) are utilized to reduce computational costs

In pursuit of the most appropriate modelling strategy, significant effort should be always spent on model validation

Thank you
for your attention