SAND2022-11972 C

Sandia National Laboratories

Exceptional service in the national interest

Coupled Fluid Structure Interaction Simulation on a Horizontal Tidal Current Turbine

Verification & Validation of CFD Model and Demonstration of Coupled Fluid Structure Interaction Model

Dongyoung Kim, Budi Gunawan

UMERC+METS 2022 Conference

September 13-14, 2022

Portland, OR

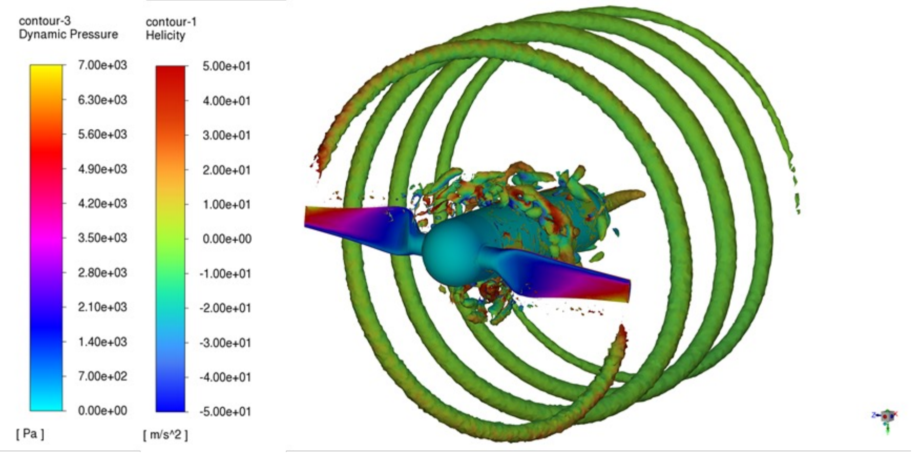
Sandia National Laboratories is a multimission laboratory managed and operated by National Technology and Engineering Solutions of Sandia LLC, a wholly owned subsidiary of Honeywell International Inc. for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

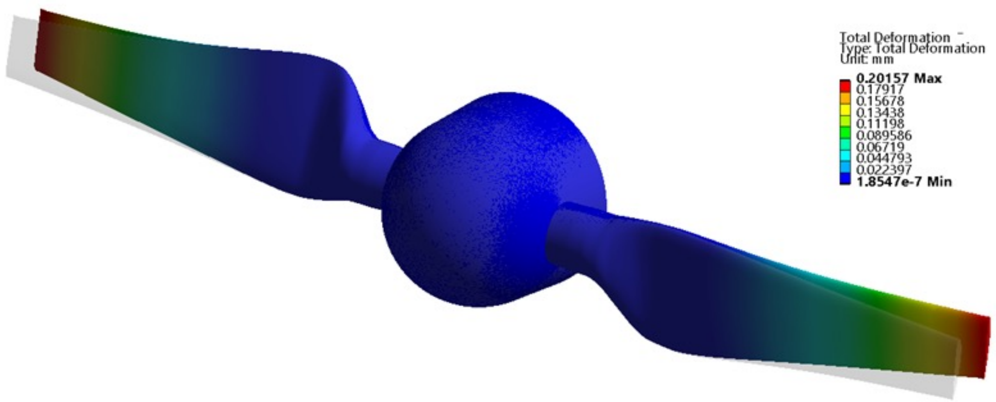
Introduction

- Turbine blades experience a complete cycle of reversed stress during each evolution
  - Deflections (deformations) on the blade during its operation

- Conventional design studies on tidal current turbines
  - Computational Fluid Dynamics (CFD) with a simple rigid blade assumption
  - Finite Element Analysis (FEA) with simplified hydrodynamic loads from low-fidelity methods

FSI model will yield time-accurate solutions for loading and performance of a deforming rotor*



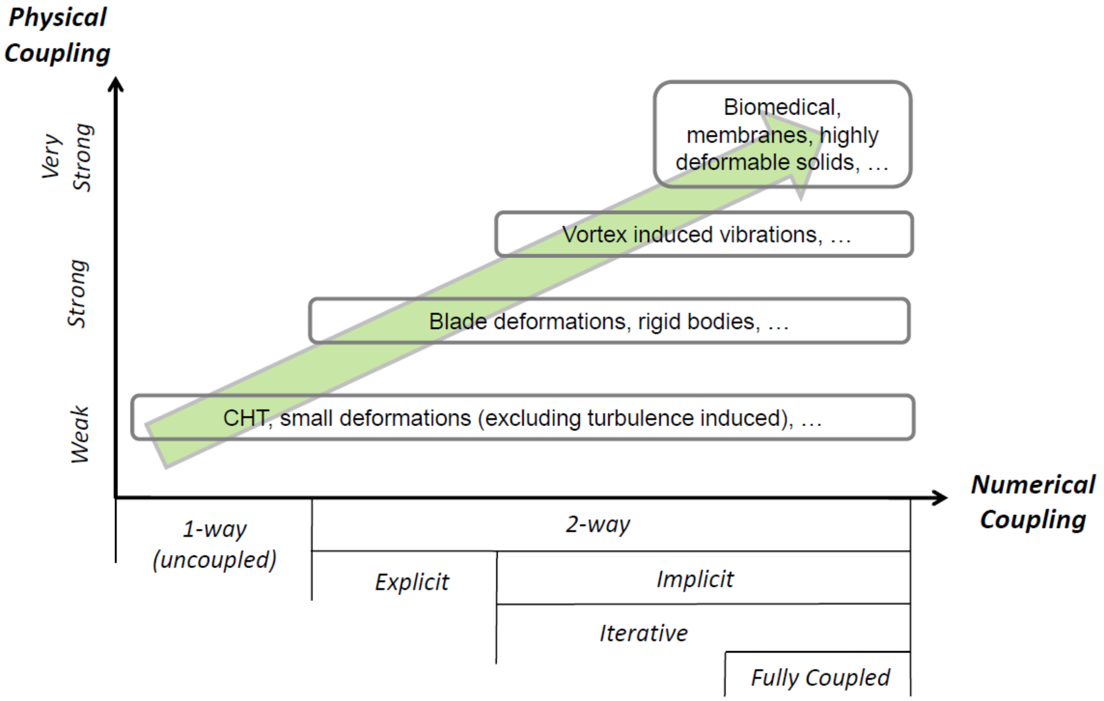


Q-criterion iso-surface colored by helicity and pressure contour on the turbine surface (left) and total deformation of blades (right)

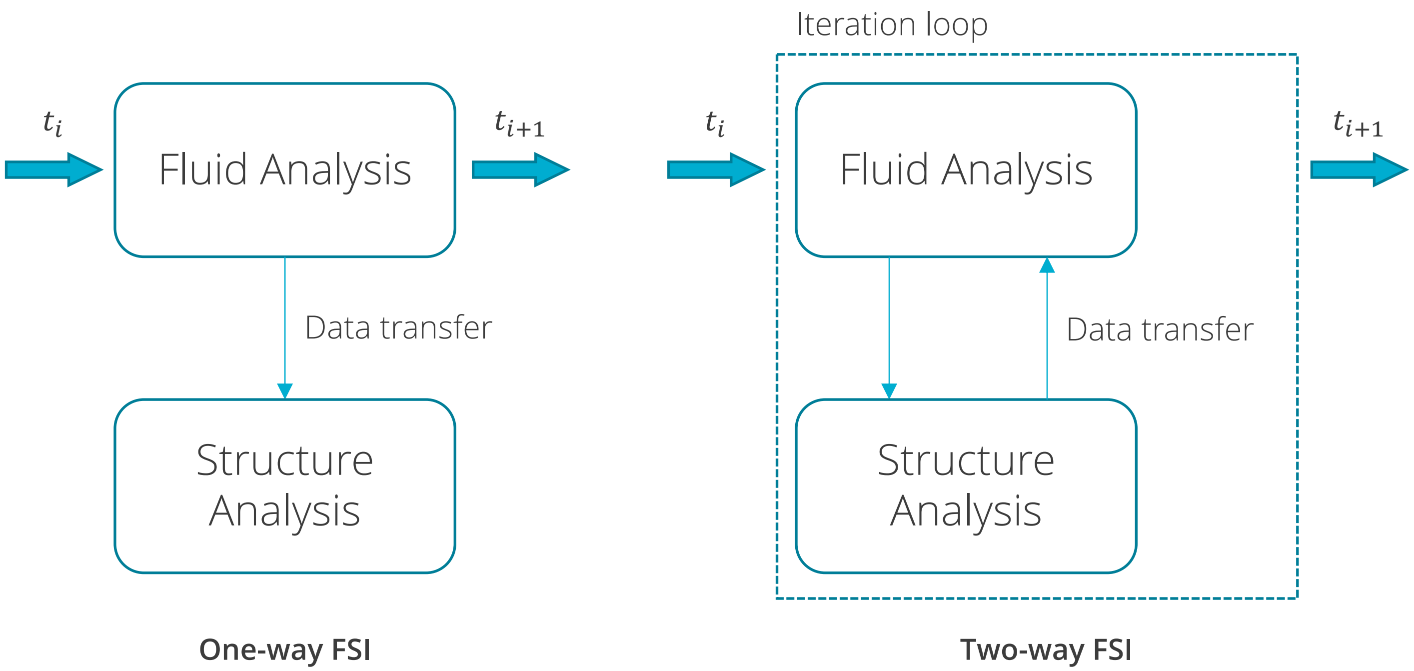
*Daniel L. Laird, Erick L. Johnson, Margaret E. Ochs, and Blake Boren, Technological Cost-Reduction Pathways for Axial-Flow Turbines in the Marine Hydrokinetic Environment, SANDIA REPORT, SAND2013-7203 (2013).

Introduction

- Fluid Structure Interaction (FSI)
  - Categorized by the degree of physical coupling between CFD and FEA solvers



The degree of physical coupling and numerical coupling approaches



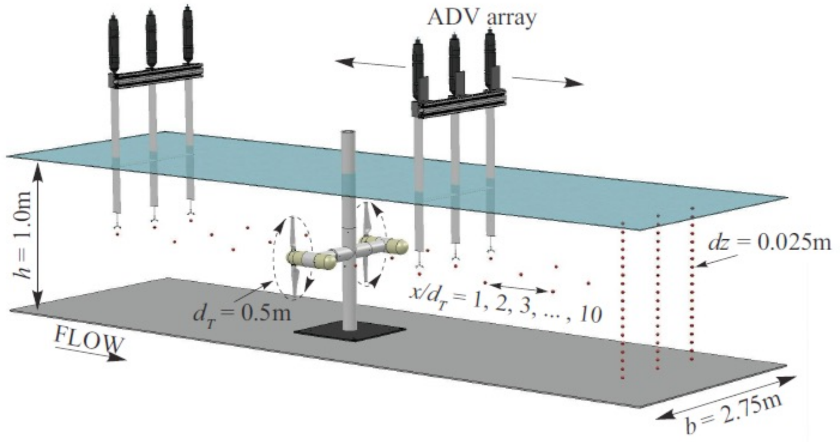
ANSYS Fluent and Mechanical are used for FSI modeling

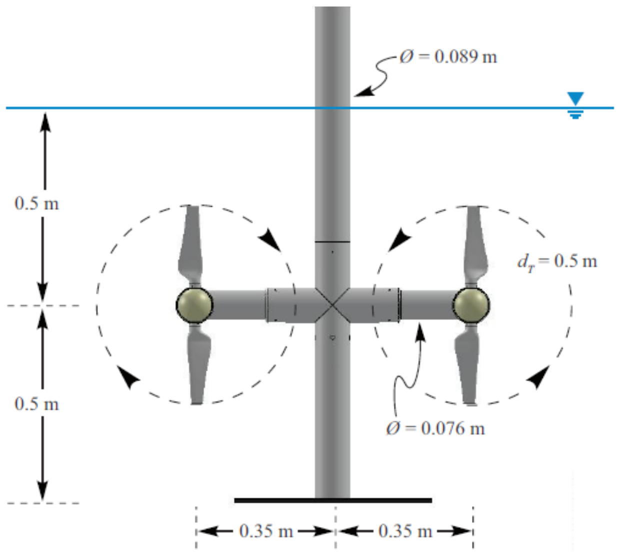
Reference Model 1 (RM1)

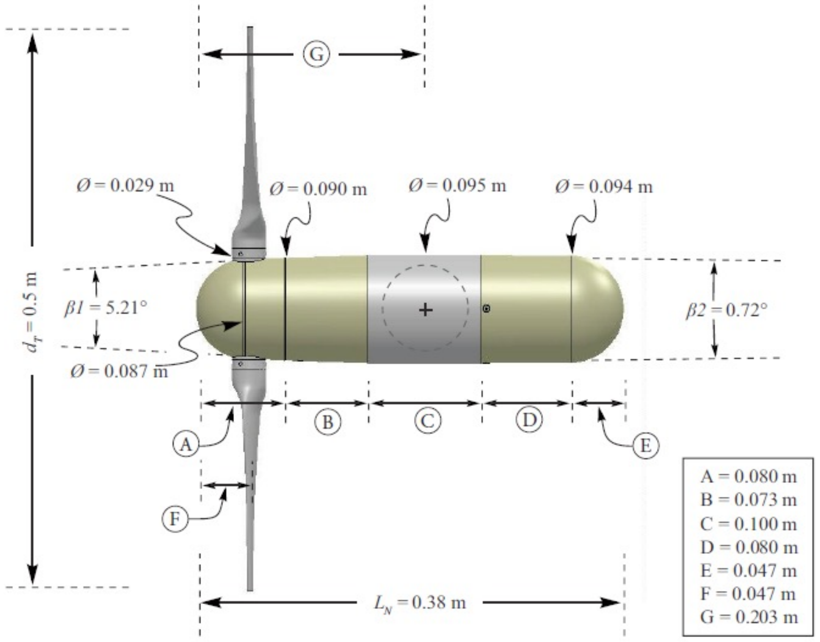
Geometric characteristics of RM1

- 1:40 scale model tested at St. Anthony Falls Laboratory (SAFL) (Hill et al., 2014 & 2020)

Parameter	1:40 Model
Q_w	2.425 m^3s^{-1}
h	1.0 m
T_{H2O}	18.0–20.5 °C
u_{hub}	1.04 ms^{-1}
Fr	0.28
Re_c at C_{p-Opt}	3.1×10^5
Re_D	4.4×10^5
$NACA$	4415
d_T	0.5 m
h_{hub}	0.5 m
σ	13.7%
η	14.3%
λ	1 to 9



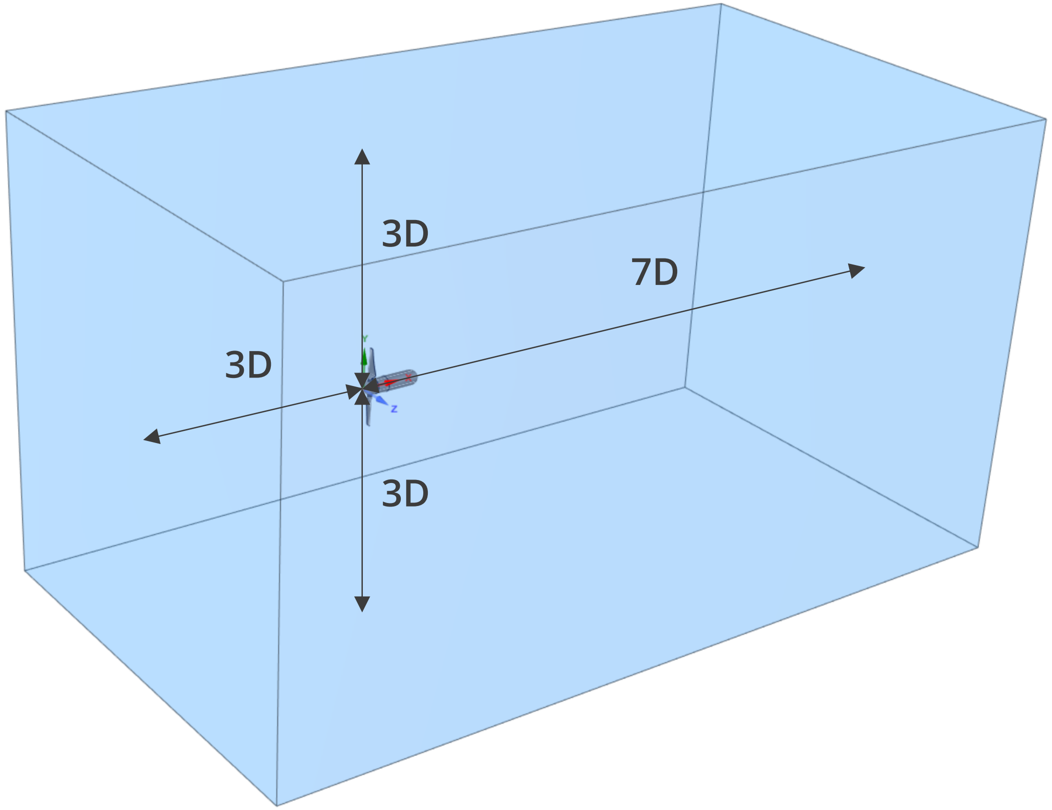




CFD Setup

Computational domain and boundary conditions (w/o blockage)

- One rotor only
- Cut off 1.5% of chord length for mesh quality
- Blockage effect is ignored
- (0, 0, 0,) at the nose of the rotor
- Inlet: 1.04 m/s uniform flow
- Outlet: zero gauge pressure
- Symmetry: on top, bottom, and sides
- No-slip wall: on rotor and nacelle

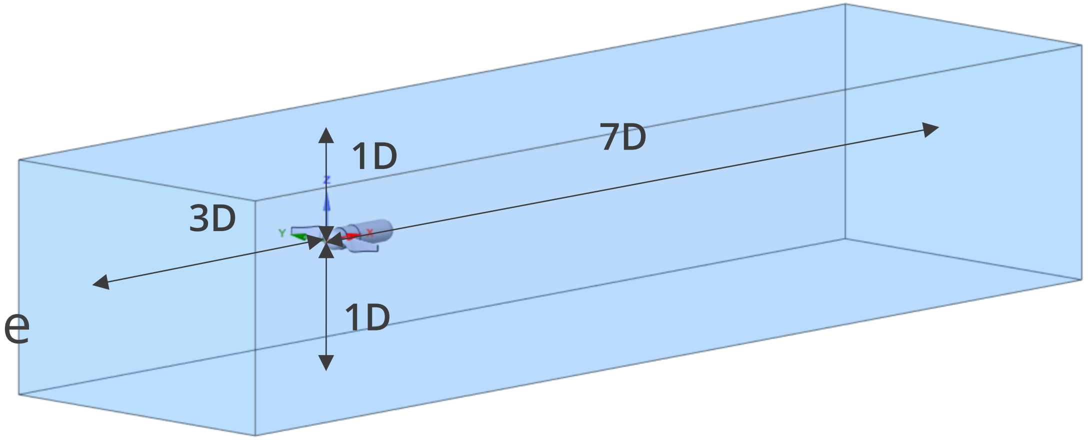


Computational domain for the simulation without blockage effect

CFD Setup

Computational domain and boundary conditions (w/ blockage)

- One rotor only
- Cut off 1.5% of chord length for mesh quality
- Blockage is applied to bottom and side wall
- (0, 0, 0,) at the nose of the rotor
- Inlet: fully developed turbulent flow velocity profile
- Outlet: zero gauge pressure
- Symmetry: left side
- No-slip wall: rotor, nacelle, bottom and right side
- Free surface effect is ignored (Slip wall)

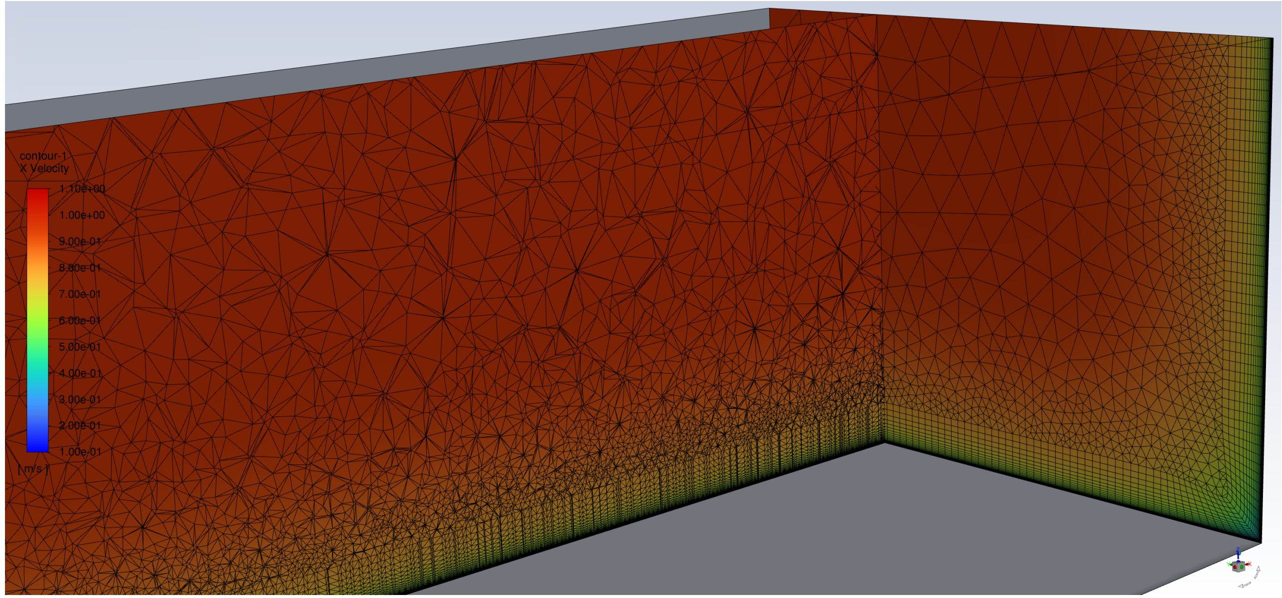


Computational domain for the simulation with blockage effect

CFD Setup

Inlet boundary condition (w/ blockage)

- Water tunnel simulation
  - To obtain velocity profile of fully developed turbulent flow
  - 40m (80D) long - The RM1 model was located 40m downstream of the baffles
  - No-slip wall BC on bottom and right side
  - Volumetric flow rate, $Q_w = 2.425\ m^3/s$



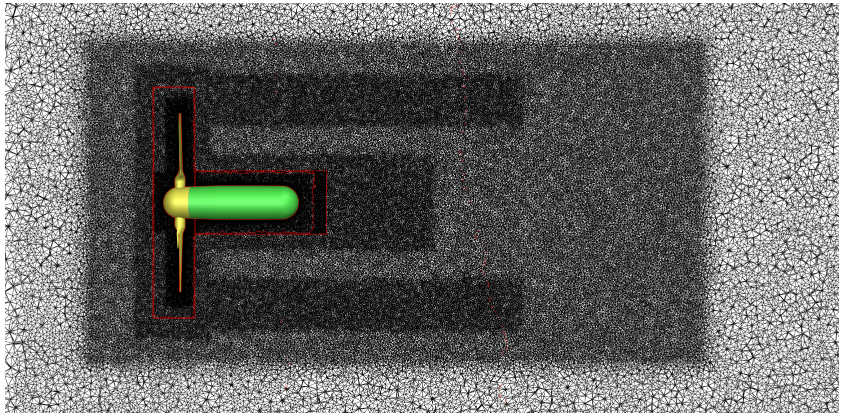
Velocity contour of fully developed turbulent flow from the water tunnel simulation

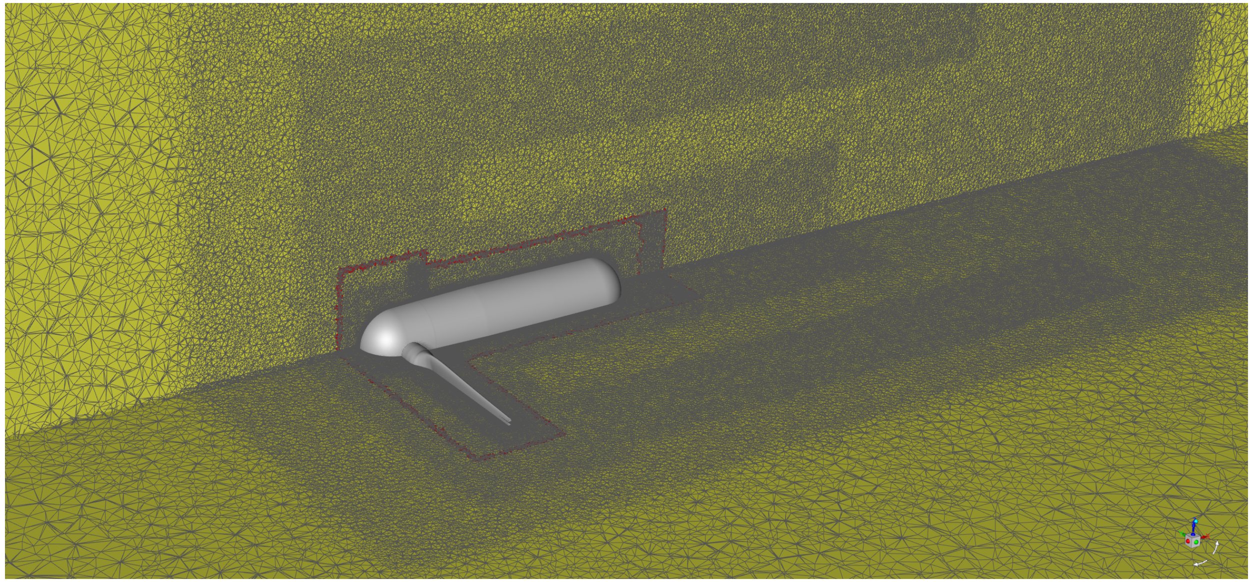
CFD Setup

Computational Mesh (Medium grid)

- Tetrahedral mesh with overset multi-blocks (# of cells)
  - Rotor: 9.3M
  - Nacelle: 1.1M
  - Bkg w/ refinement: 19.1M (w/ blockage)
  - Total: 29.4M

- Prism layers on the rotor and nacelle wall
  - $y^+ = 1.4$ ($\Delta y = 3.44\times10^{-5}\ m$)
  - Growth rate: 1.2
  - Total number of layers: 20
    - $h_{total} = 9.1\times10^{-3}\ m, \delta_{turb.est.} = 1.8\times10^{-3}\ m$





Computational Mesh for rotor and nacelle overset blocks and background domain
(Donor cells for overset method are colored by red)

CFD Setup

Mathematical Model and Numerical Method

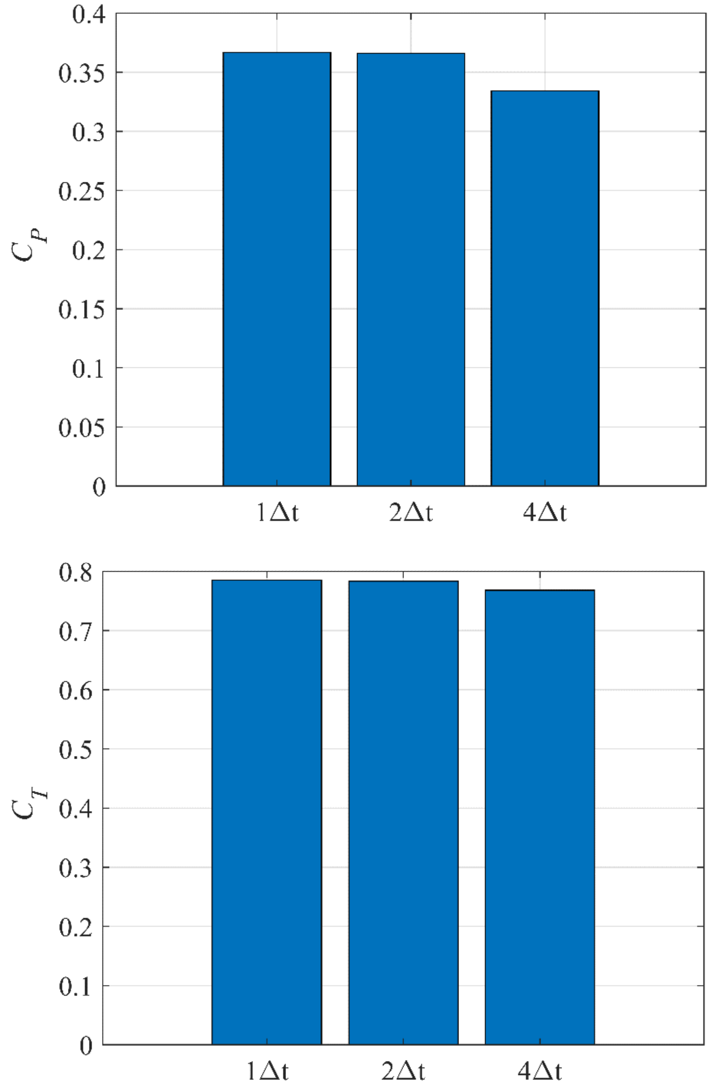
- Viscous model:
  - SST k-omega model
- Pressure-velocity coupling:
  - Pressure-based coupled solver
- Spatial discretization:
  - Pressure: second order
  - Momentum: second order upwind
  - Turbulence model: second order upwind
- Temporal discretization:
  - Transient formulation: first order implicit

Temporal Convergence

Timestep size dependency (w/o blockage)

	Time step size	C_P (diff, %)	C_T (diff, %)
N_1	$1°$ rotation per Δt	0.3667 (-)	0.7850 (-)
N_2	$2°$ rotation per Δt	0.3660 (0.20)	0.7833 (0.22)
N_3	$4°$ rotation per Δt	0.3343 (8.86)	0.7681 (2.15)
U_{k_1}		0.008%	0.054%

U_{k_1} is uncertainty of N_1 obtained from the method of Stern et al. (2006); and Xing and Stern (2010)



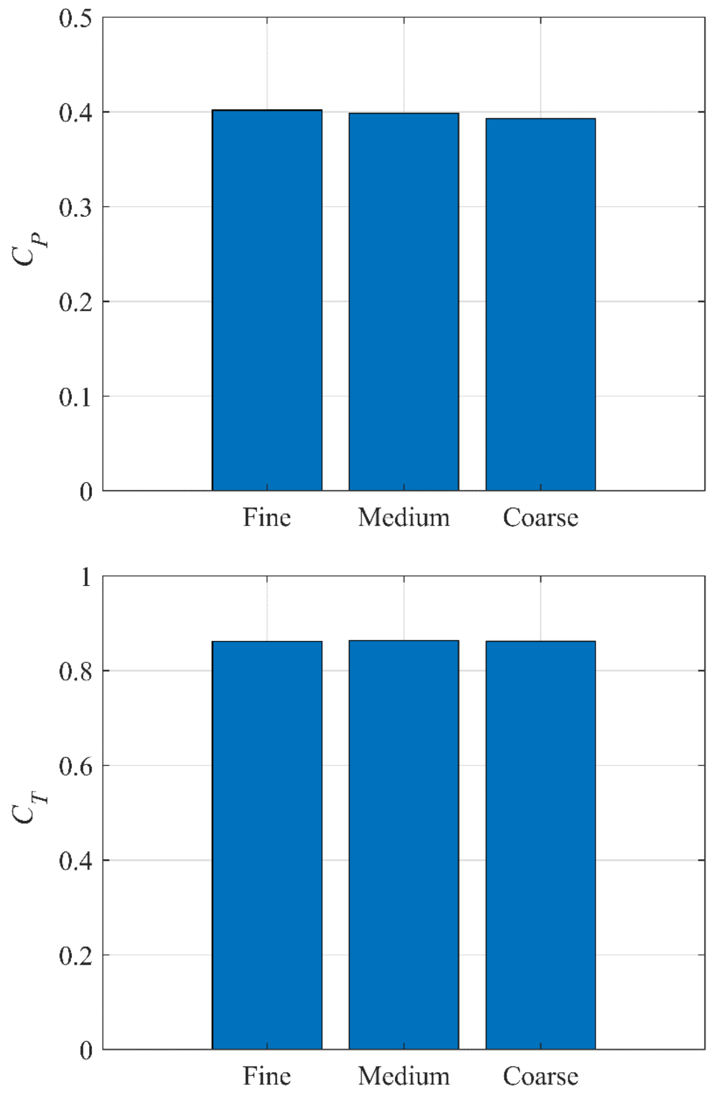
Estimated C_P and C_T depends on the time step size

Spatial Convergence

Mesh size dependency study (w/ blockage)

	# of cells	C_P (diff, %)	C_T (diff, %)
G_1	$66.2\ M\ (y^+ = 1)$	0.4018 (-)	0.8617 (-)
G_2	$29.4\ M\ (y^+ = 1.4)$	0.3984 (0.83)	0.8632 (0.18)
G_3	$14.5\ M\ (y^+ = 2)$	0.3928 (2.24)	0.8622 (0.06)
U_{k_1}		1.007%	-

U_{k_1} is uncertainty of G_1 obtained from the method of Stern et al. (2006); and Xing and Stern (2010)

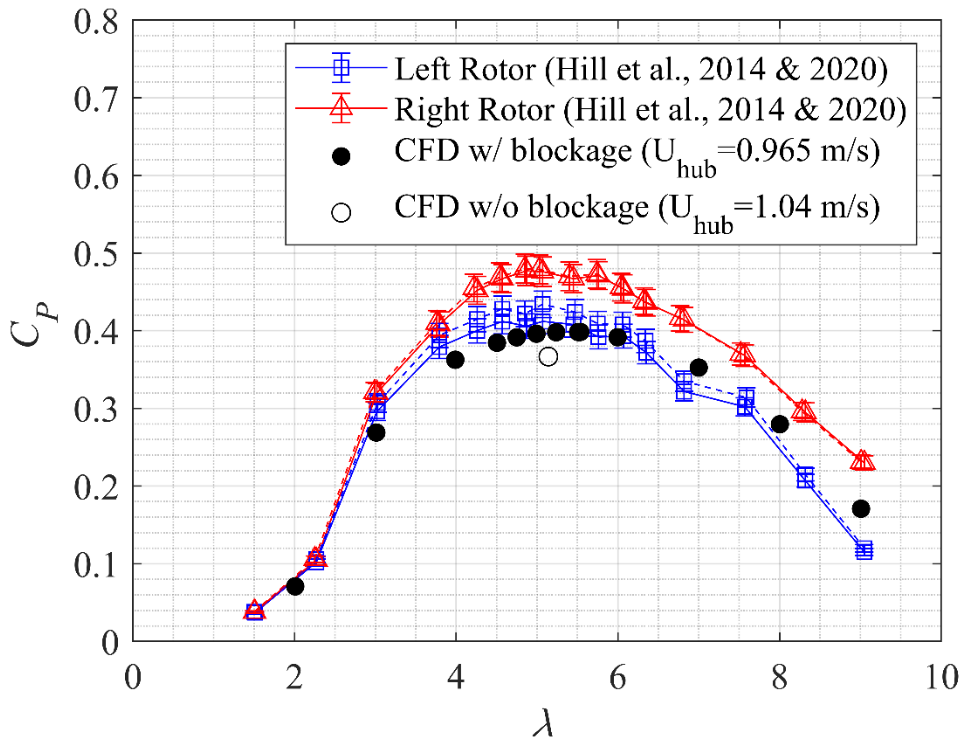


Estimated C_P and C_T depends on the mesh size

Turbine Performance

Coefficient of power

- Discrepancy between CFD w/o blockage and Exp. (Hill et al, 2014 & 2020) results due to the extensive blockage effect (14.3%)



Measured and estimated C_P vs. λ (coefficient of power vs. tip-speed ratio). Solid and dashed lines are from Hill et al, 2014 and 2020, respectively)

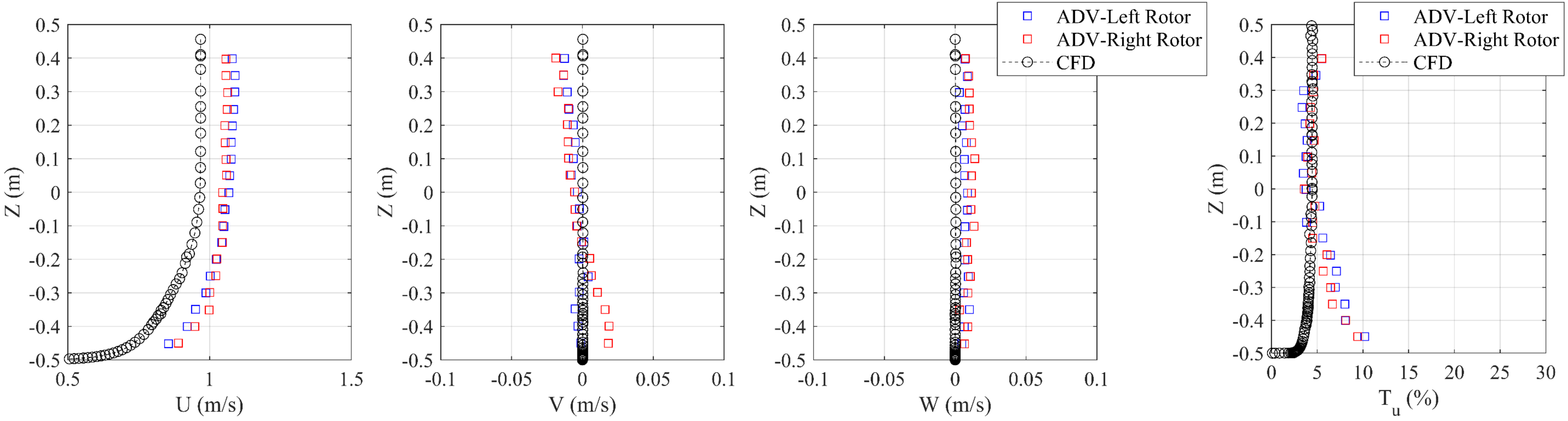
@ 204 rpm	C_P	*Uncertainty*
Exp. Left Rotor (TSR = 5.07)	0.412, 0.434	3.9 %
Exp. Right Rotor (TSR = 5.03)	0.476, 0.479	
CFD w/o blockage (TSR = 5.14)	0.367	
CFD w/ blockage (TSR = 5.54)	0.402	

Inflow Characteristics

Velocity and turbulence intensity profiles

- $U_{\text{hub,Exp}} \approx 1.04\, m/s$ @ $x = -3d_T$
- $U_{\text{hub,CFD}} \approx 0.965\, m/s$ @ $x = -3d_T$

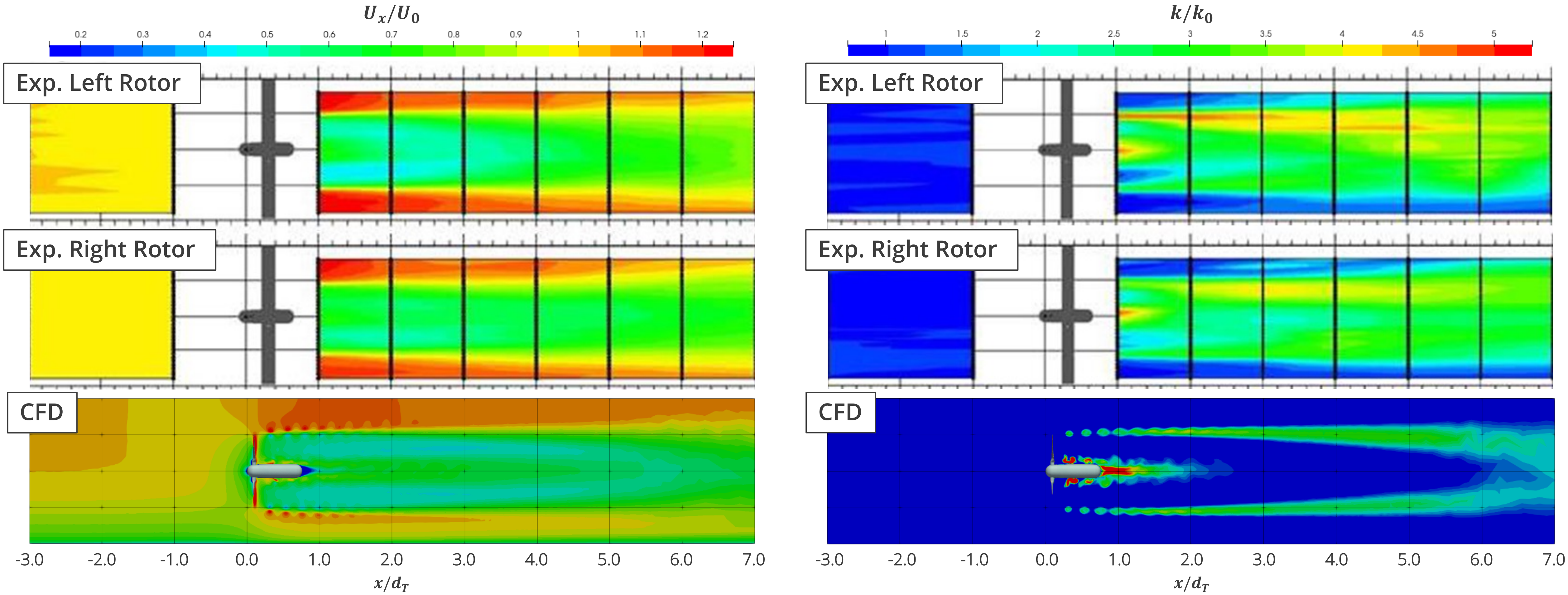
d_T: Turbine diameter
T_u: Turbulence intensity
ADV: Acoustic Doppler Velocimetry



Measured (red and blue square) and estimated (black circle) profiles for velocity components and turbulence intensity

Turbine Wake Characteristics

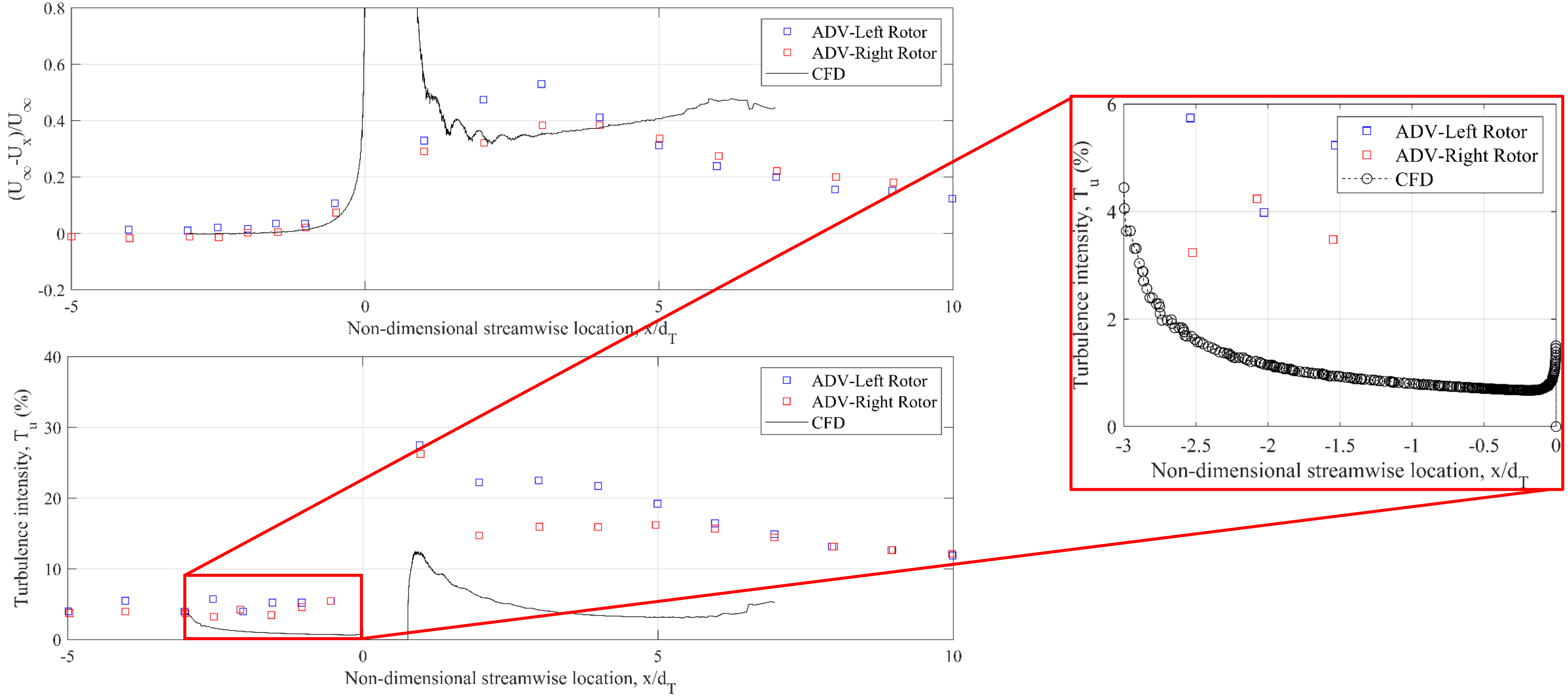
Normalized streamwise velocity and turbulent kinetic energy



Measured (top) and estimated (bottom) normalized streamwise velocity (left column) and turbulent kinetic energy (right column) in x-z plane

Turbine Wake Characteristics

Streamwise velocity deficit and turbulence intensity

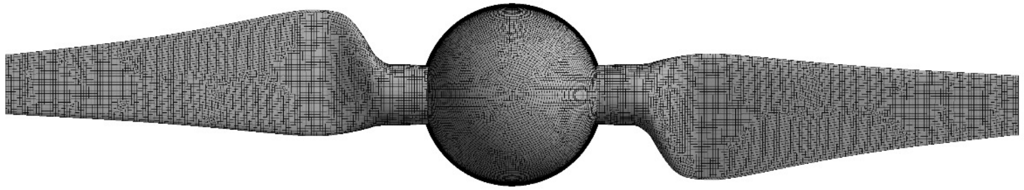


Measured and estimated velocity deficit (top) and turbulence intensity (bottom) at hub height along the streamwise direction

FEA Setup

Geometry and mesh

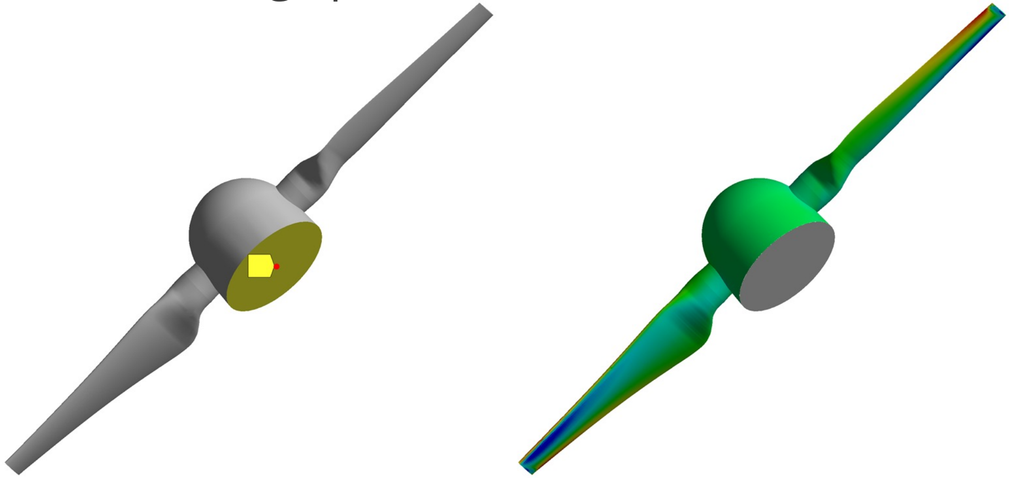
- Rotor only
- Hexahedral mesh with quadratic element order
- Modelled as a solid made from aluminum alloy



Generated mesh for FEA simulation (# of elements = 1.1M)

Boundary conditions

- Assigned angular velocity corresponding to the turbine rotating speed
- Displacement support at the turbine hub center
- A fluid-solid interface on the rotor surface

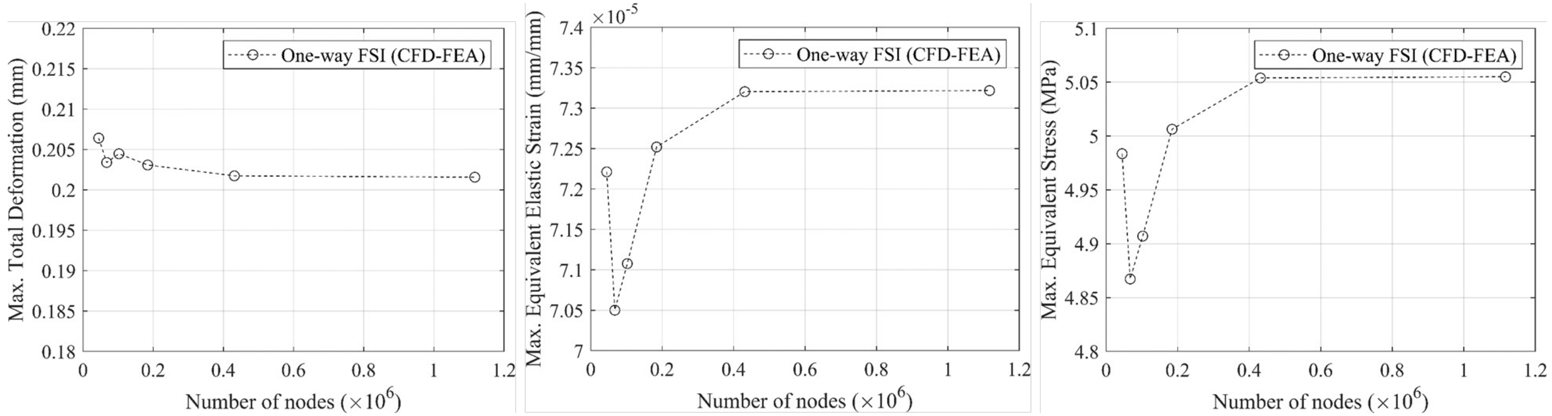


Remote displacement point (left) and pressure on the fluid-solid interface (right)

One-way FSI

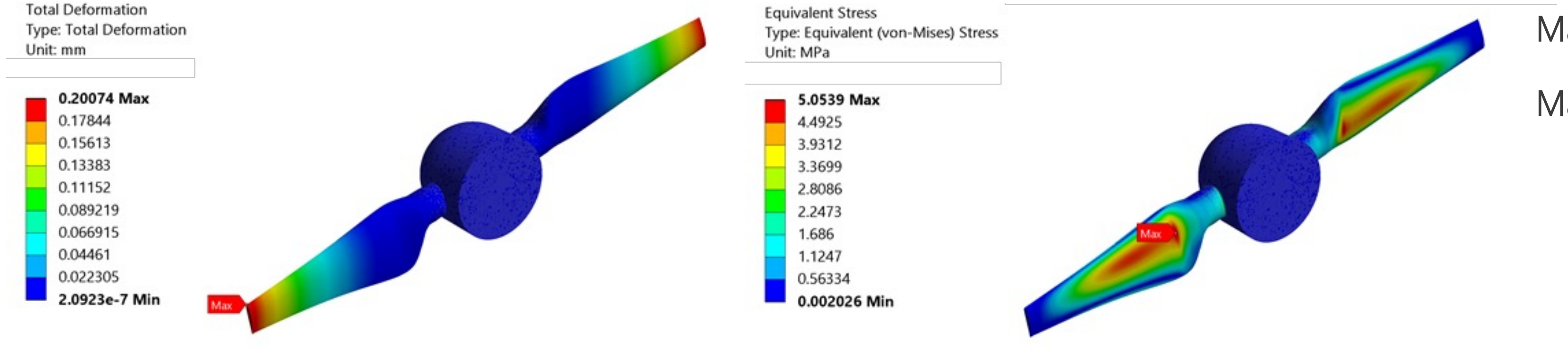
Simulation results at 204 rpm (TSR = 5.5)

- Mesh size dependency



Variation of estimated maximum deformation (left), strain (middle), and stress (right) with mesh density

- Estimated total deformation and equivalent stress



Max. total deformation = 0.2 mm

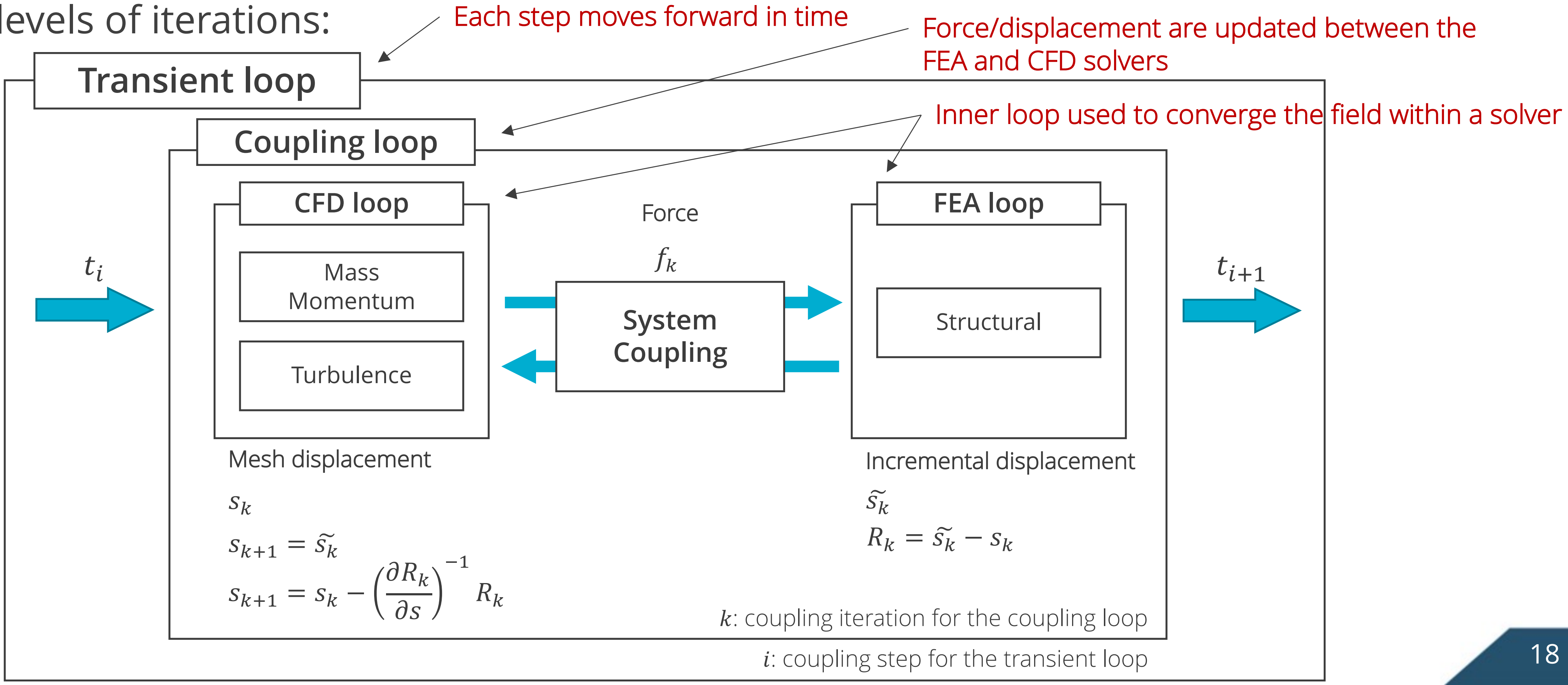
Max. equivalent stress = 5.05 MPa

Instantaneous contour plots of total deformation (left) and equivalent stress (right) on rotor

Two-way FSI Model

Two-way iteratively implicit approach

- Iterate within each time step to obtain an implicit solution
- Three levels of iterations:



Conclusion and Further Works

- Transient two-way coupled FSI is modeled based on CFD and one-way FSI models
- Require constrain method for freestream turbulence intensity in CFD
- Decision of suitable timestep for two-way FSI to avoid negative cell volume from mesh smoothing
- Investigation of the influence of blade deformation on hydrodynamic parameters
- Evaluation of LCOE of full scale turbine made from composite materials

Thank you