Design and Optimization of Contra-Rotating Fans

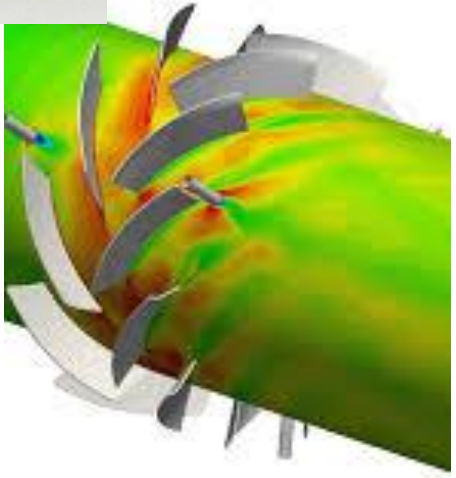
Oliver Velde[#], Christian Friebe[&], Marius Korfanty[#]

CFturbo GmbH

& ILK Institut für Luft- und Kältetechnik gGmbH

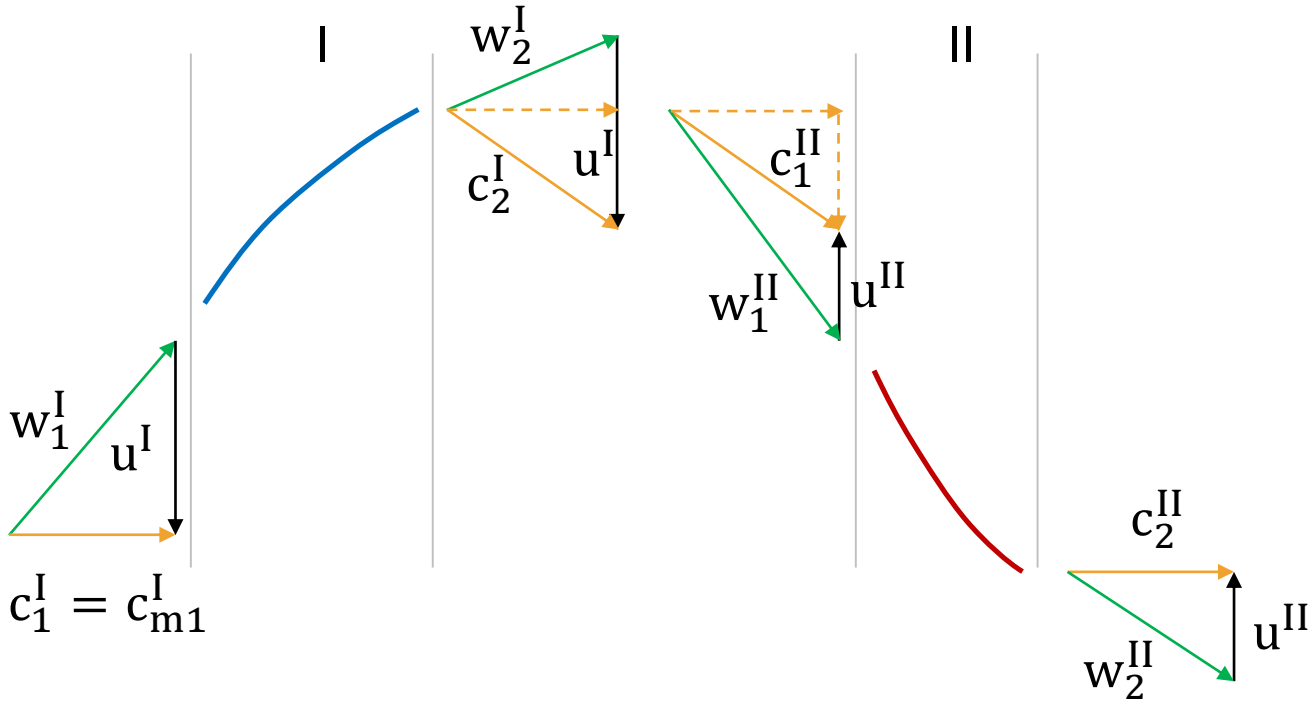
Motivation to use Contra-rotating fans

- Very compact compared to a two stage fan with guiding vanes
- Smaller impeller diameter and/or lower speed compared to single stage fans
- Alternative for low pressure radial ventilators
- Potential for better acoustic behavior



Contra-rotating fan principle

First impeller generates a **negative** pre-swirl for the second impeller

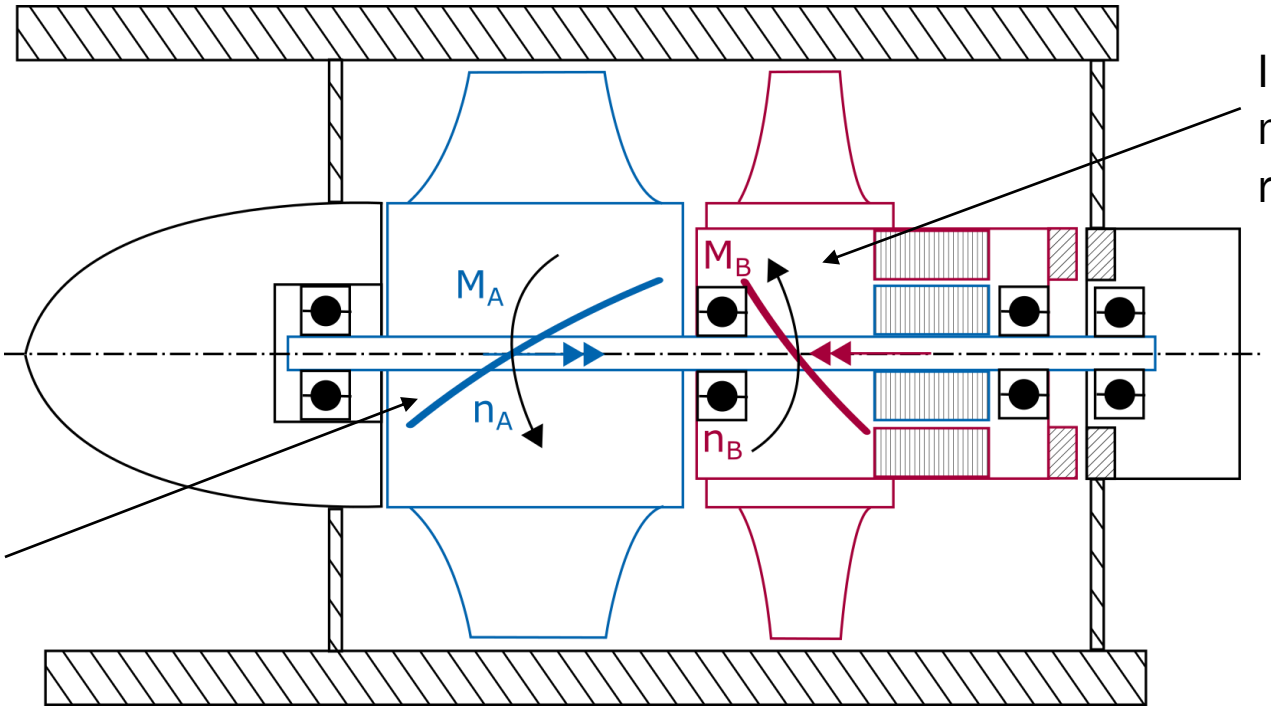


$$Y_{Euler} = \Delta(c_u \cdot u) \rightarrow \; Y_I = c_{u2}^I \cdot u^I, \qquad Y_{II} = c_{u2}^{II} \cdot u^{II}$$

$$Y_I + Y_{II} = c_{u2}^I \cdot \left(u^I + u^{II}\right)$$

Gear less single motor driven Contra-rotating fan

Torque/Power distribution imposed by aerodynamic design of impellers



Contra-rotating fan design

Design point: pressure difference Δp_{tot}, flow Q, Fluid properties

Power distribution between 2 impellers defined by either of

- Pressure difference and speed: $\Delta p_{tot,I}$, n_I und $\Delta p_{tot,II}$, n_{II}
- Torque and speed: T_I, n_I und T_{II}, n_{II} (with $T \cdot n = \Delta p_{tot} \cdot Q$)

Example:

	fan	impeller I	impeller II
Q [m^3/h]	540	540	540
Δp_{tot} [Pa]	1000	690	310
n [rpm]		8000	-5000

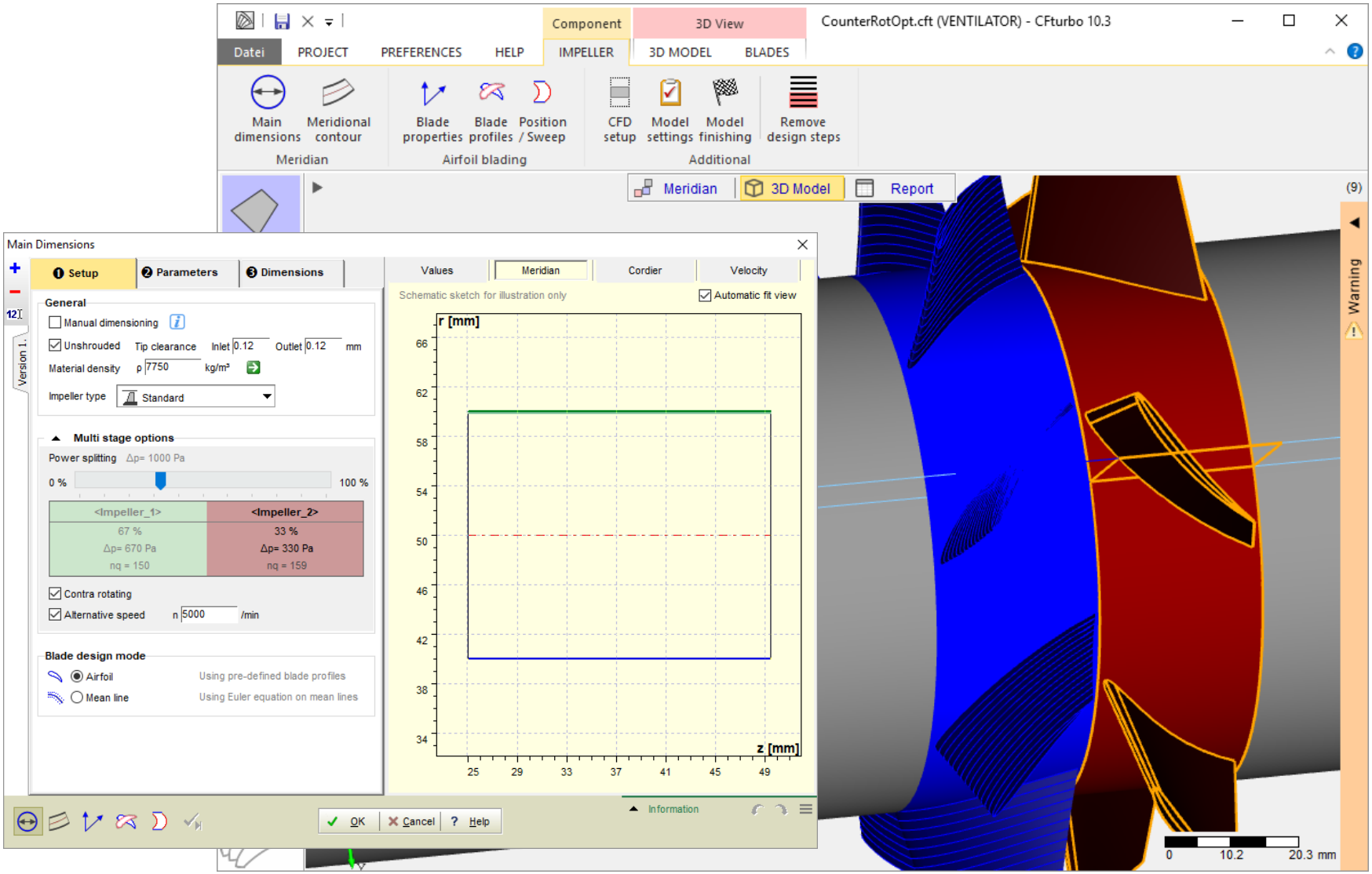
Design Point

Initial design

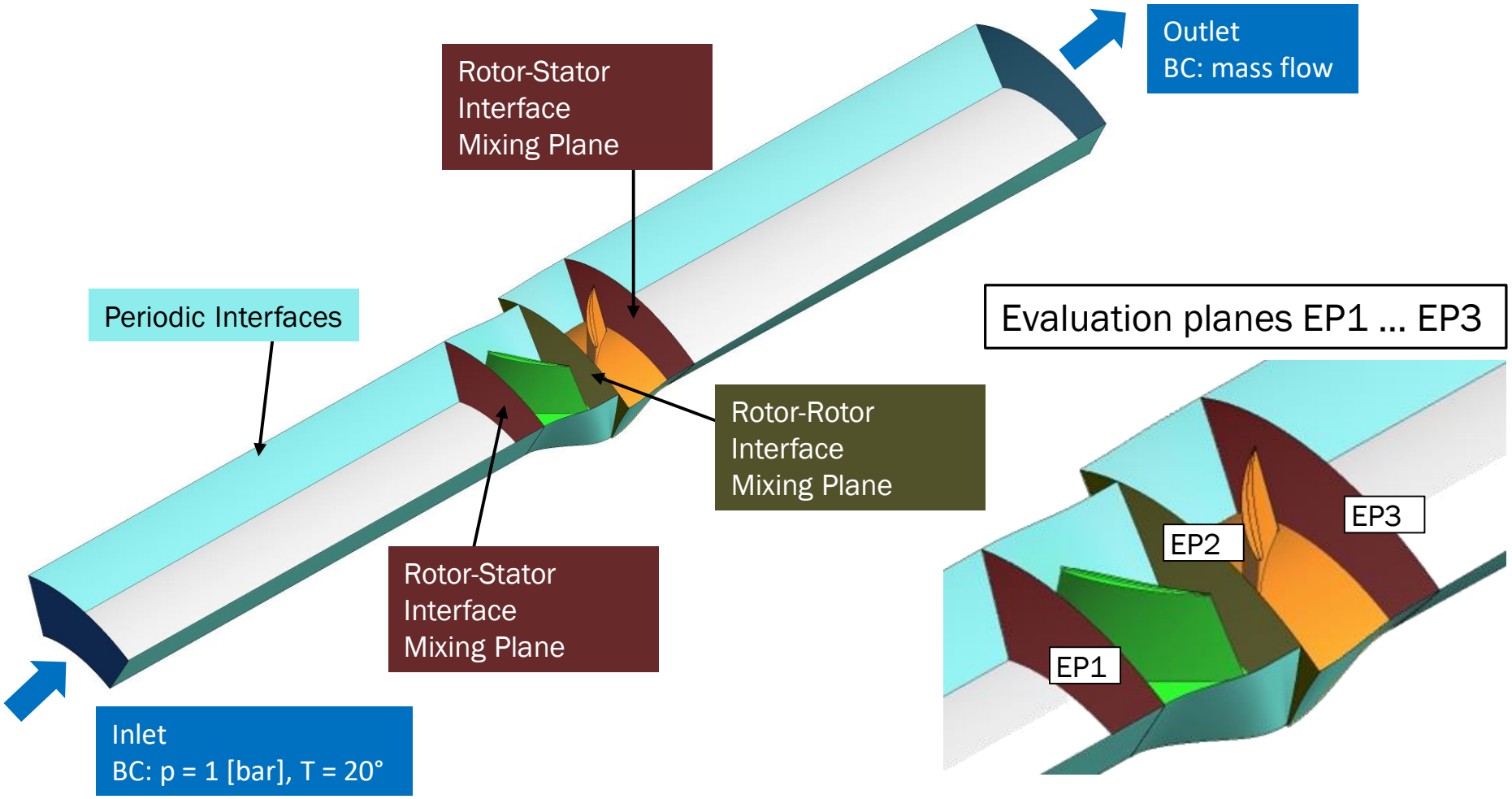
Initial design:

Number of blades [-]	**8**	**6**
Profile	NACA 6508	NACA 6508
Stagger angle λ [°]	46.7 .. 36.1	45.0 .. 36.3
Chord length l [mm]	32 .. 24	32 .. 26

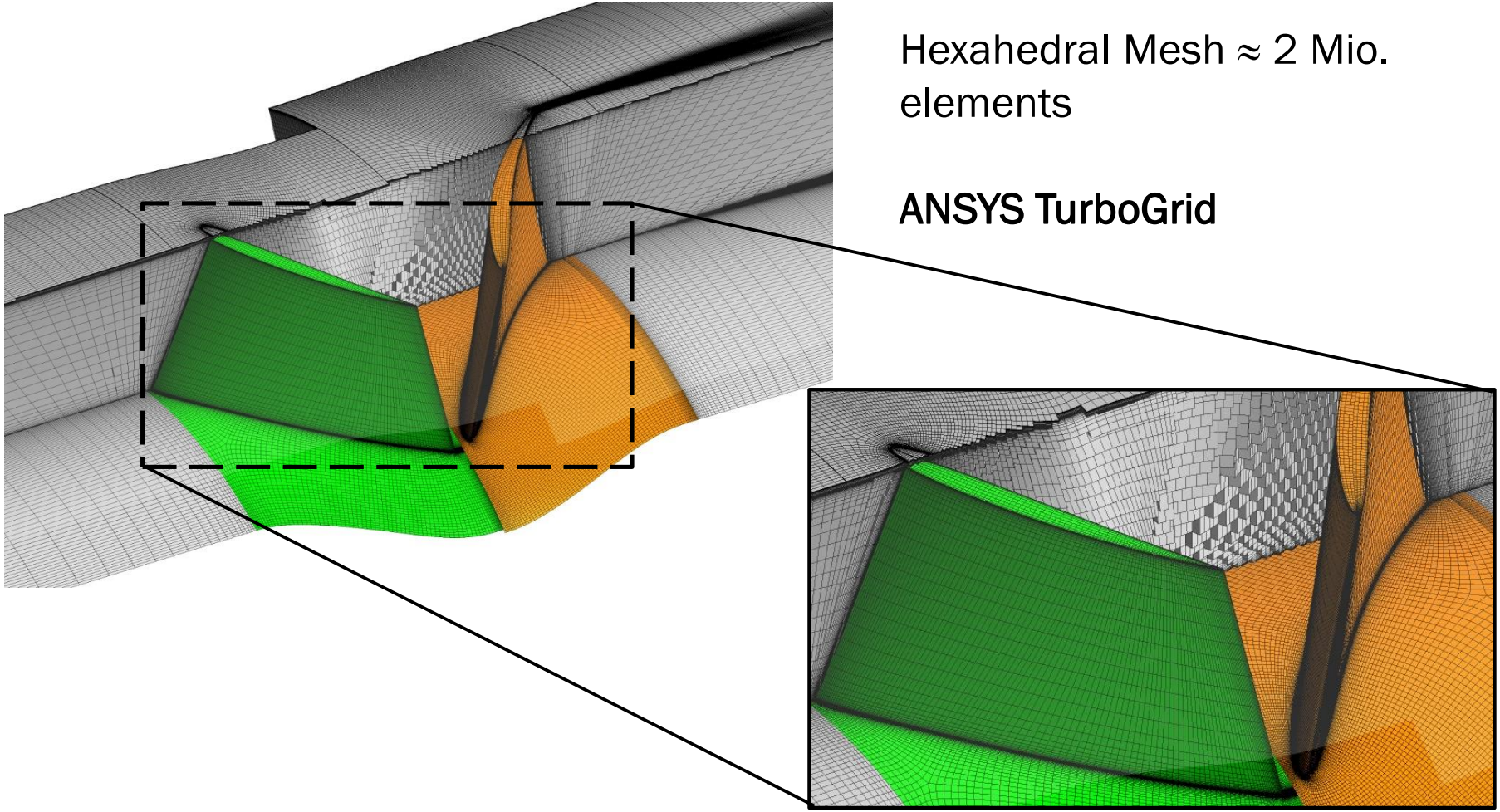
Contra-rotating fan design with CFturbo



Computational model



Mesh



Hexahedral Mesh ≈ 2 Mio. elements

ANSYS TurboGrid

Boundary conditions & solver settings

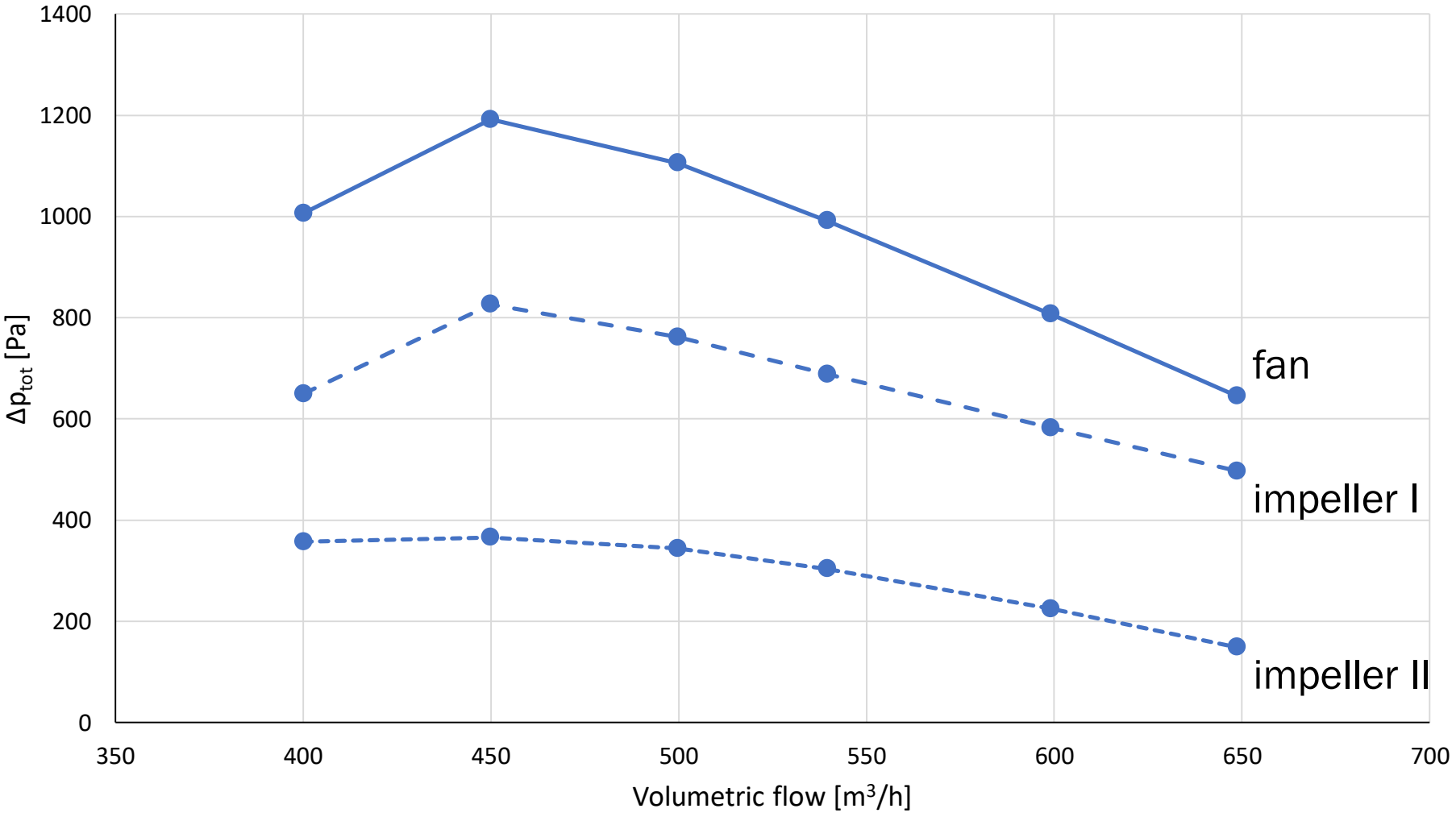
- INLET Pressure: 1 [bar]
- OUTLET Volumetric flow: 400 … 650 [m^3/h]

- Rotational speed: 8000 rpm for impeller I
  -5000 rpm for impeller II
- Fluid properties: Air (Perfect Gas)

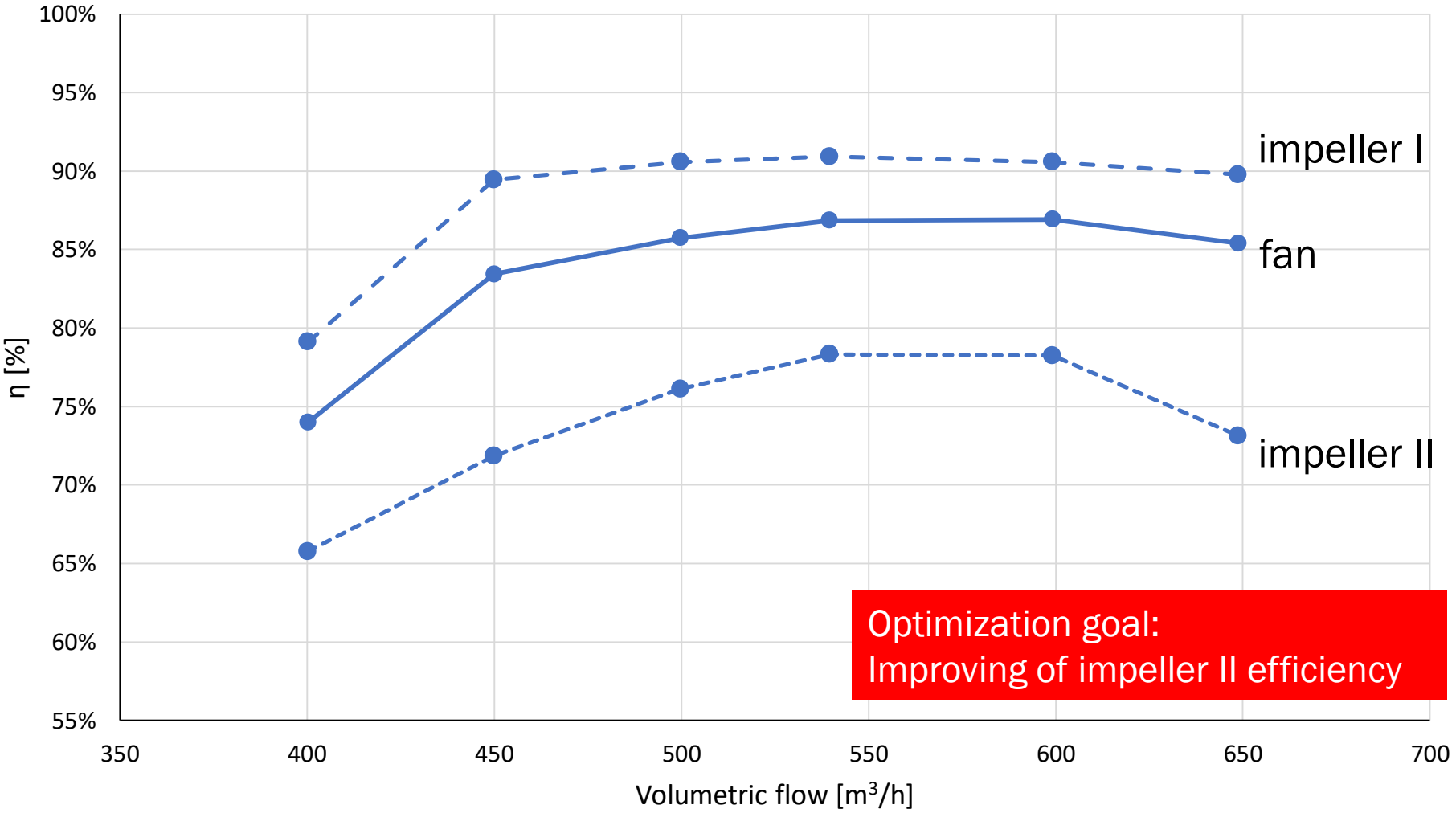
ANSYS-CFX v18.0

- Steady State Simulation
- Heat Transport (Total Energy)
- Mixing-Plane Interface
- SST-Turbulence model
- High Resolution differencing scheme

Simulation of initial design, results



Simulation of initial design, results



Optimization of initial design

Optimization process:

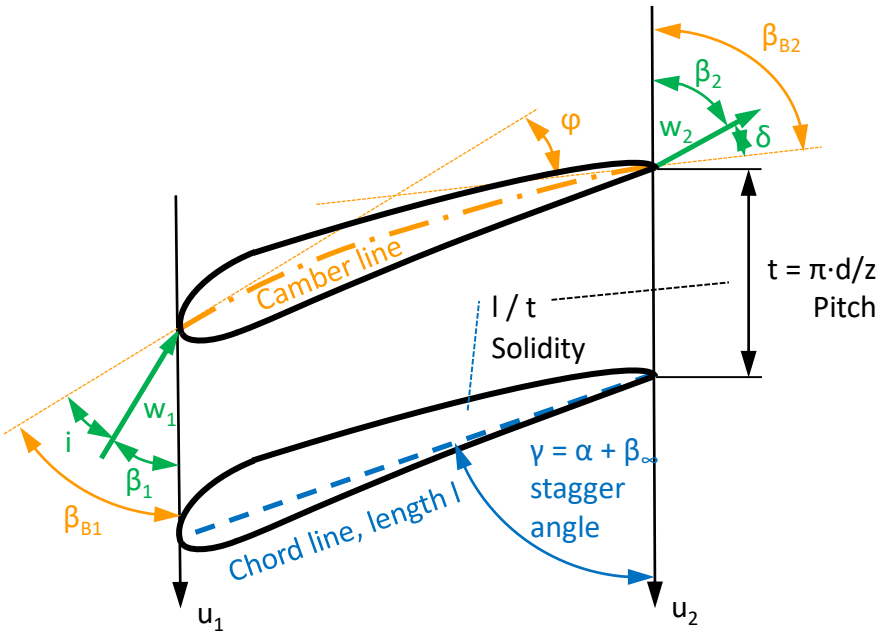
Control unit for complete process

Optimization of initial design

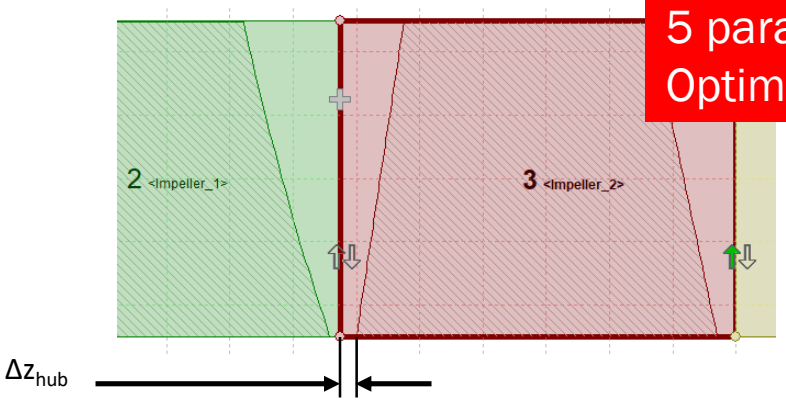
Objective: improve second impeller with respect to efficiency

Restriction: match design point

Parameters: only impeller II

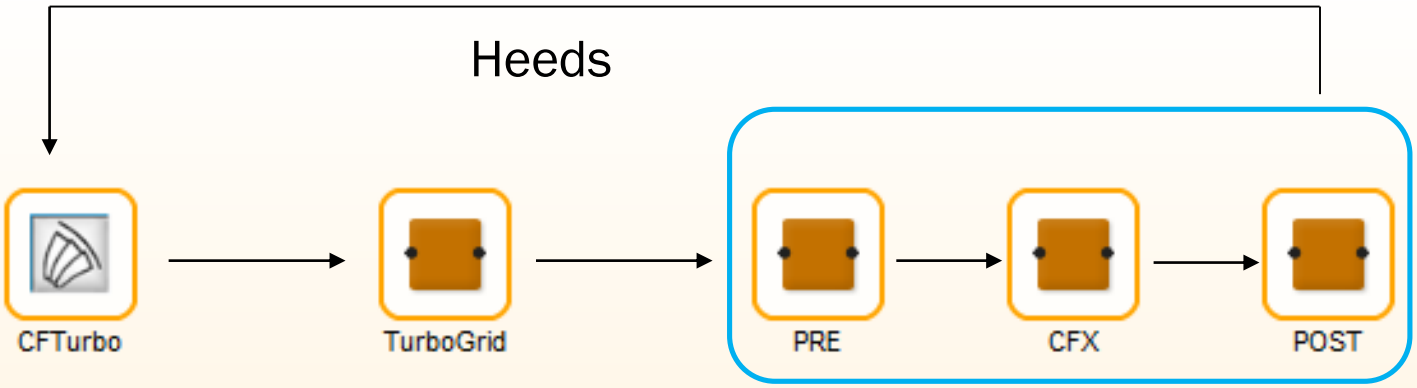


Parameters	range
Number of blades z [-]	4 .. 10
Stagger angle λ_{hub} [°]	20 .. 60
Stagger angle λ_{tip} [°]	20 .. 60
Chord length l_{hub}[mm]	32 (const.)
Chord length l_{tip}[mm]	20 .. 32
Leading edge position Δz_{hub} [mm]	1 .. 8



5 parameters for Optimization

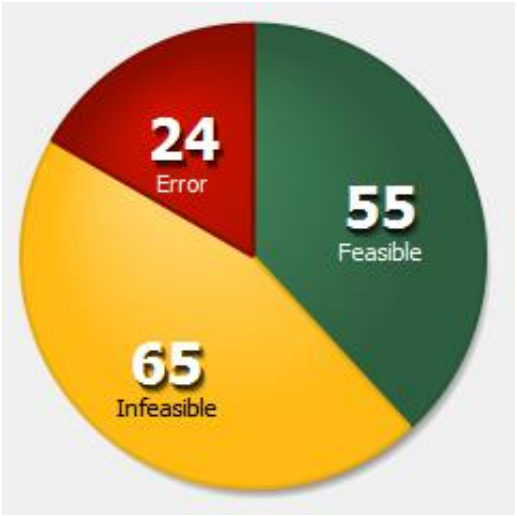
Optimization process



CFturbo:
Parametric CAD model

ANSYS TurboGrid:
Mesh generation

ANSYS CFX:
Simulation & Postprocessing



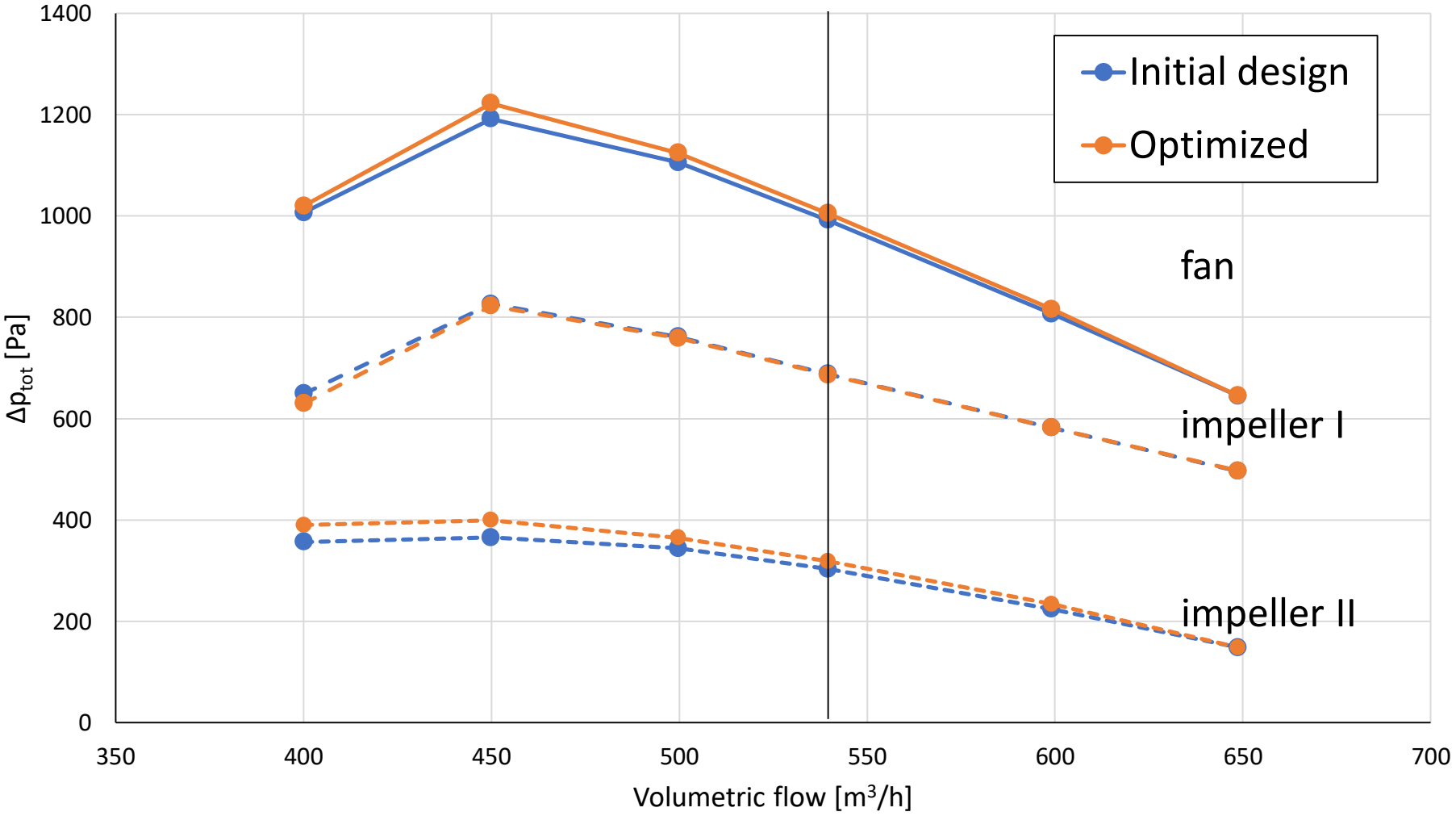
144 Designs
70min/Design / 8 Processors

Optimization results impeller II

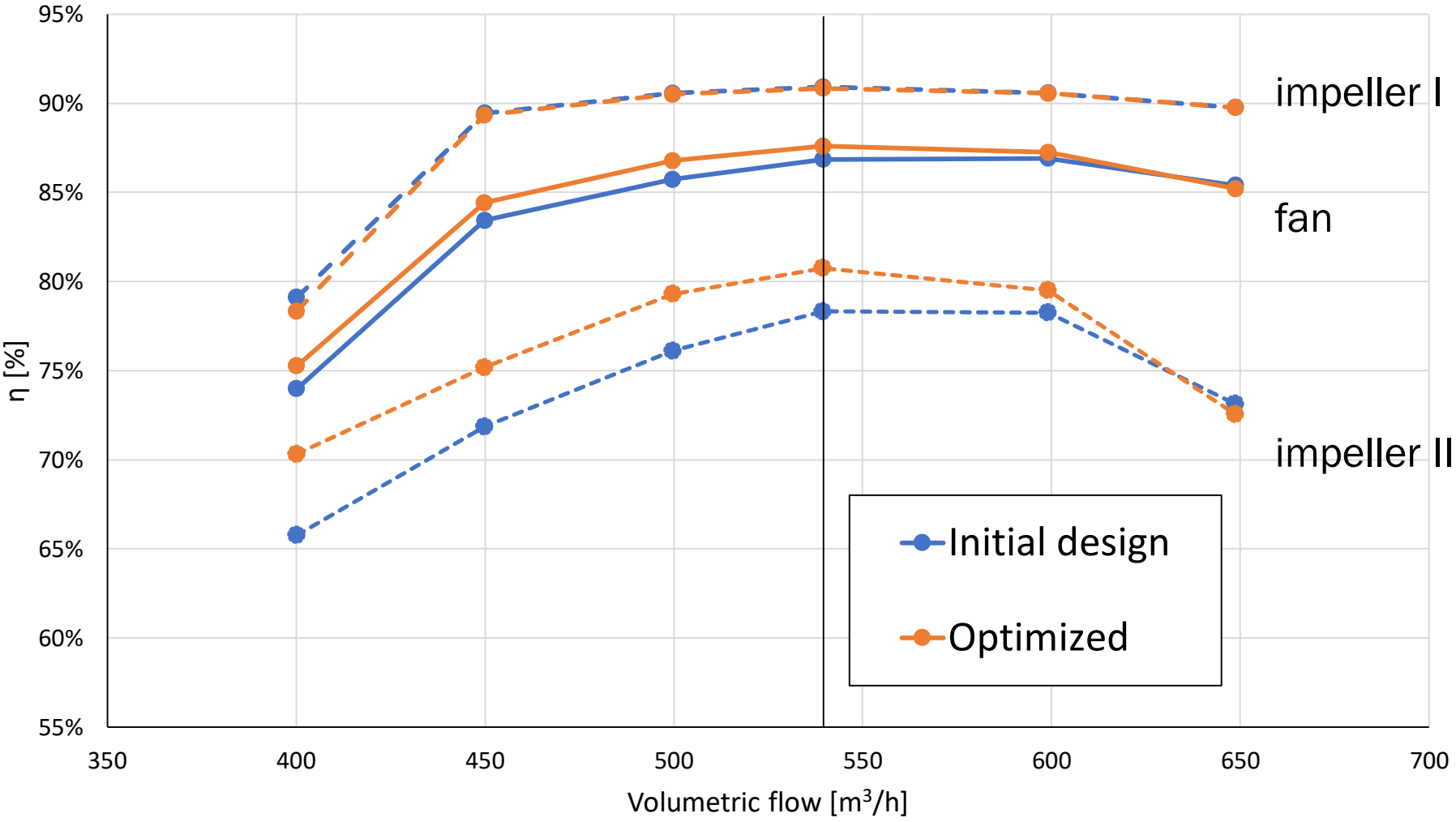
	Initial design impeller II	Optimized design impeller II
Total pressure difference Δp_{tot} [Pa]	303	319
Efficiency η [%]	78	81

	Initial design impeller II	Optimized design impeller II
Number of blades z [-]	6	7
Stagger angle λ_{hub} [°]	45.0	50.0
Stagger angle λ_{tip} [°]	36.1	30.4
Chord length l_{hub}[mm]	32	32
Chord length l_{tip}[mm]	26	26
Leading edge position Δz_{hub} [mm]	1	2.6

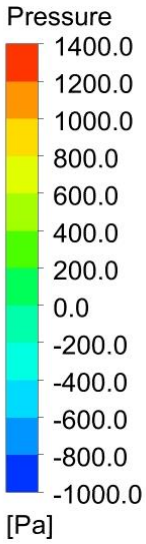
Performance curve

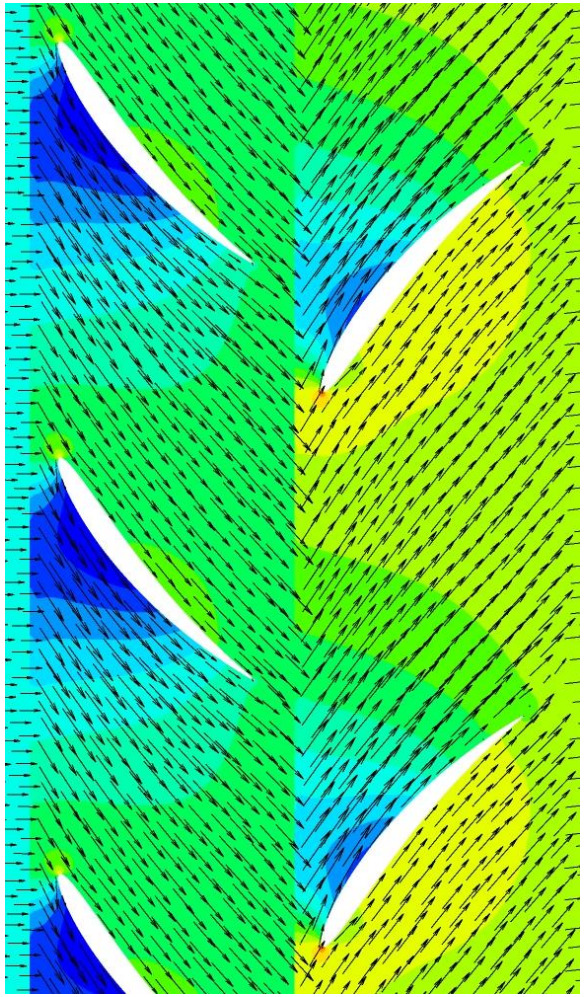


Efficiency

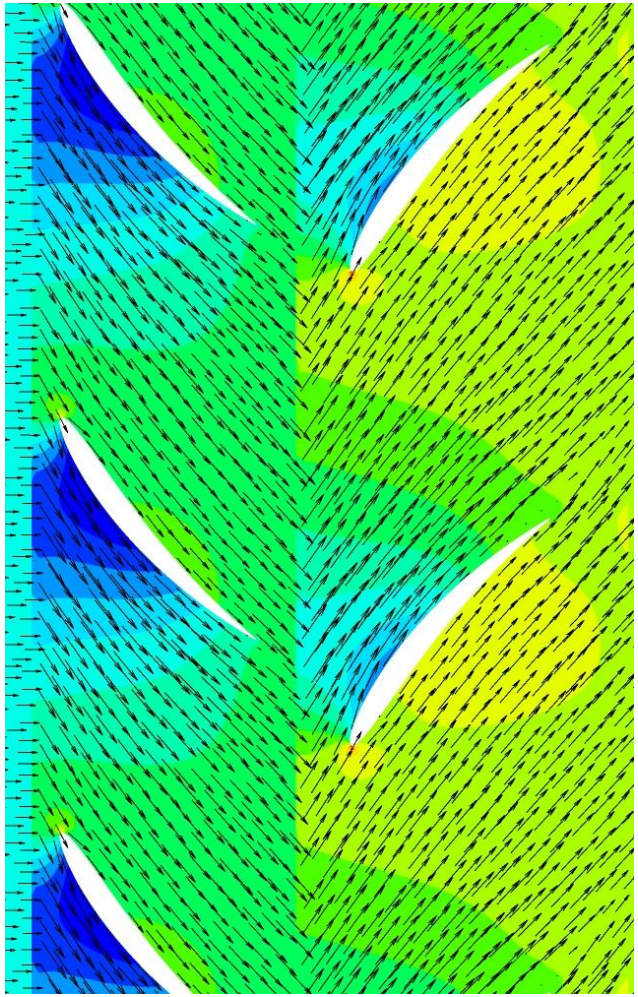

95%
90%
85%
80%
75%
70%
65%
60%
55%
η [%]
350
400
450
500
550
600
650
700
Volumetric flow [m³/h]
impeller I
fan
impeller II
Initial design
Optimized

Pressure distribution mid span



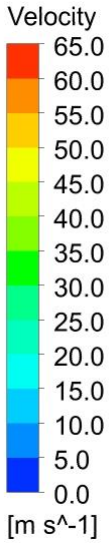


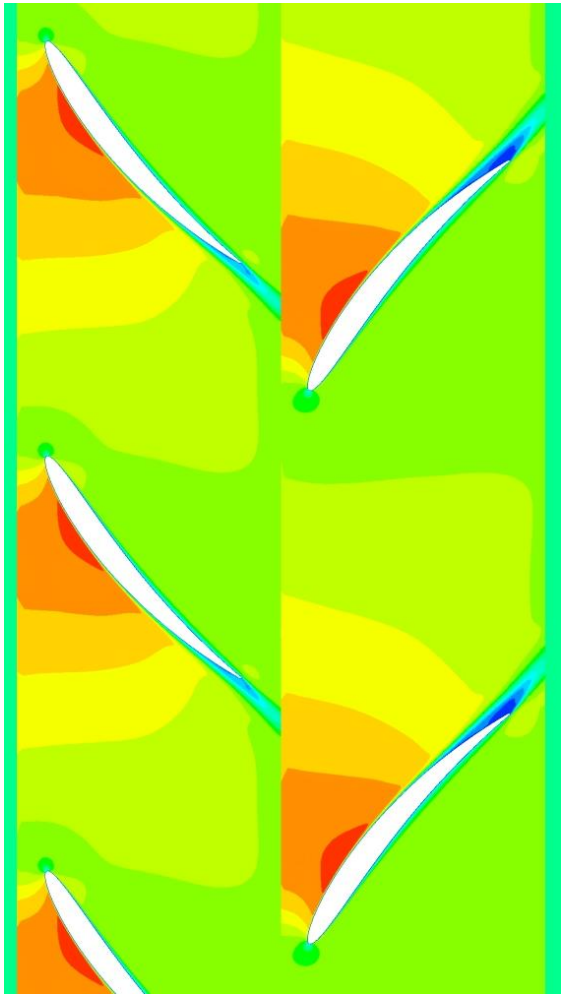
Initial design



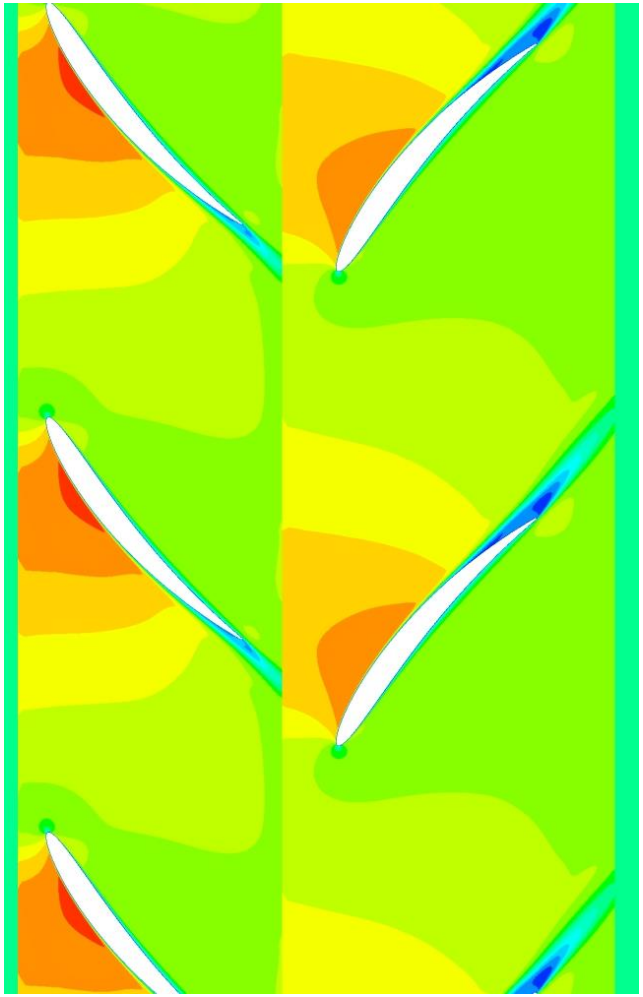
Optimized

Relative velocity distribution mid span





Initial design



Optimized

Summary

- Automated process for design and optimization for Contra-rotating fans
- Reasonable initial design for both impellers using CFturbo software
- Optimization of second impeller only
- Simple method: passage only, steady state simulation
- Computational resources
- Validation in progress
  - Transient flow simulations/360°
  - More detailed geometry modeling
  - Experiments
- Acoustic investigations planned