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

\[ Y_{Euler} = \Delta (c_u \cdot u) \rightarrow Y_I = c^{I}_{u2} \cdot u^I \quad , \quad Y_{II} = c^{II}_{u2} \cdot u^{II} \]

\[ Y_I + Y_{II} = c^{I}_{u2} \cdot (u^I + u^{II}) \]
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 $\Delta p_{\text{tot}}$, flow $Q$, Fluid properties

Power distribution between 2 impellers defined by either of
- Pressure difference and speed: $\Delta p_{\text{tot},\text{I}}$, $n_{\text{I}}$ und $\Delta p_{\text{tot},\text{II}}$, $n_{\text{II}}$
- Torque and speed: $T_{\text{I}}$, $n_{\text{I}}$ und $T_{\text{II}}$, $n_{\text{II}}$ (with $T \cdot n = \Delta p_{\text{tot}} \cdot Q$)

Example:

<table>
<thead>
<tr>
<th></th>
<th>fan</th>
<th>impeller I</th>
<th>impeller II</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Q$ [m$^3$/h]</td>
<td>540</td>
<td>540</td>
<td>540</td>
</tr>
<tr>
<td>$\Delta p_{\text{tot}}$ [Pa]</td>
<td>1000</td>
<td>690</td>
<td>310</td>
</tr>
<tr>
<td>$n$ [rpm]</td>
<td>8000</td>
<td>8000</td>
<td>-5000</td>
</tr>
</tbody>
</table>

Initial design:

<table>
<thead>
<tr>
<th></th>
<th>Number of blades [-]</th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Profile</td>
<td>NACA 6508</td>
<td>NACA 6508</td>
<td></td>
</tr>
<tr>
<td>Stagger angle $\lambda$ [°]</td>
<td>46.7 .. 36.1</td>
<td>45.0 .. 36.3</td>
<td></td>
</tr>
<tr>
<td>Chord length $l$ [mm]</td>
<td>32 .. 24</td>
<td>32 .. 26</td>
<td></td>
</tr>
</tbody>
</table>
Contra-rotating fan design with CFturbo
Computational model

Inlet BC: \( p = 1 \) [bar], \( T = 20^\circ \)

Outlet BC: mass flow

Periodic Interfaces

Evaluation planes EP1 ... EP3

Rotor-Stator Interface Mixing Plane

Rotor-Rotor Interface Mixing Plane
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

Graph showing the relationship between volumetric flow and total pressure drop for different components:
- Fan
- Impeller I
- Impeller II

The x-axis represents volumetric flow in m$^3$/h, and the y-axis represents total pressure drop in Pa.
Simulation of initial design, results

Optimization goal: Improving of impeller II efficiency
Optimization of initial design

Optimization process:

- Parametric CAD model
- Meshing, simulation, analysis
- Control unit for complete process

CFturbo®

Heeds
Design exploration software

ANSYS®
Optimization of initial design

**Objective:** improve second impeller with respect to efficiency

**Restriction:** match design point

**Parameters:** only impeller II

<table>
<thead>
<tr>
<th>Parameters</th>
<th>range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades $z$ [-]</td>
<td>4 .. 10</td>
</tr>
<tr>
<td>Stagger angle $\lambda_{hub}$ [°]</td>
<td>20 .. 60</td>
</tr>
<tr>
<td>Stagger angle $\lambda_{tip}$ [°]</td>
<td>20 .. 60</td>
</tr>
<tr>
<td>Chord length $l_{hub}$ [mm]</td>
<td>32 (const.)</td>
</tr>
<tr>
<td>Chord length $l_{tip}$ [mm]</td>
<td>20 .. 32</td>
</tr>
<tr>
<td>Leading edge position $\Delta z_{hub}$ [mm]</td>
<td>1 .. 8</td>
</tr>
</tbody>
</table>

5 parameters for Optimization
Optimization process

CFturb: Parametric CAD model

ANSYS TurboGrid: Mesh generation

ANSYS CFX: Simulation & Postprocessing

Heeds

144 Designs
70min/Design / 8 Processors
### Optimization results impeller II

<table>
<thead>
<tr>
<th></th>
<th>Initial design impeller II</th>
<th>Optimized design impeller II</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Total pressure difference</strong> $\Delta p_{\text{tot}}$ [Pa]</td>
<td>303</td>
<td>319</td>
</tr>
<tr>
<td><strong>Efficiency</strong> $\eta$ [%]</td>
<td>78</td>
<td>81</td>
</tr>
<tr>
<td><strong>Number of blades</strong> $z$ [-]</td>
<td>6</td>
<td>7</td>
</tr>
<tr>
<td><strong>Stagger angle</strong> $\lambda_{\text{hub}}$ [°]</td>
<td>45.0</td>
<td>50.0</td>
</tr>
<tr>
<td><strong>Stagger angle</strong> $\lambda_{\text{tip}}$ [°]</td>
<td>36.1</td>
<td>30.4</td>
</tr>
<tr>
<td><strong>Chord length</strong> $l_{\text{hub}}$ [mm]</td>
<td>32</td>
<td>32</td>
</tr>
<tr>
<td><strong>Chord length</strong> $l_{\text{tip}}$ [mm]</td>
<td>26</td>
<td>26</td>
</tr>
<tr>
<td><strong>Leading edge position</strong> $\Delta z_{\text{hub}}$ [mm]</td>
<td>1</td>
<td>2.6</td>
</tr>
</tbody>
</table>
Performance curve

![Graph](Image)

- **Initial design**
- **Optimized**

**Δp_{tot} [Pa]**

**Volumetric flow [m³/h]**

- Fan
- Impeller I
- Impeller II
Efficiency

![Graph showing efficiency vs volumetric flow for different impellers and designs.](image)

- **Initial design**
- **Optimized**

**Legend:**
- **Impeller I**
- **Impeller II**

**Axes:**
- **η [%]**
- **Volumetric flow [m³/h]**

**Data Points:**
- Various points indicating efficiency at different volumetric flows for each impeller and design.
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