Herrick Laboratories
PURDUE UNIVERSITY

2016 Purdue Conferences
Compressor Engineering
Refrigeration and Air Conditioning
High Performance Buildings

CFD SIMULATION OF A TWIN SCREW EXPANDER INCLUDING LEAKAGE FLOWS

Rainer Andres, Jan Hesse, Haris Babic, Uwe Salecker, Andreas Spille-Kohoff

CFX Berlin Software GmbH, Berlin, Germany

Alexander Nikolov, Andreas Brümmer

TU Dortmund University, Dortmund, Germany

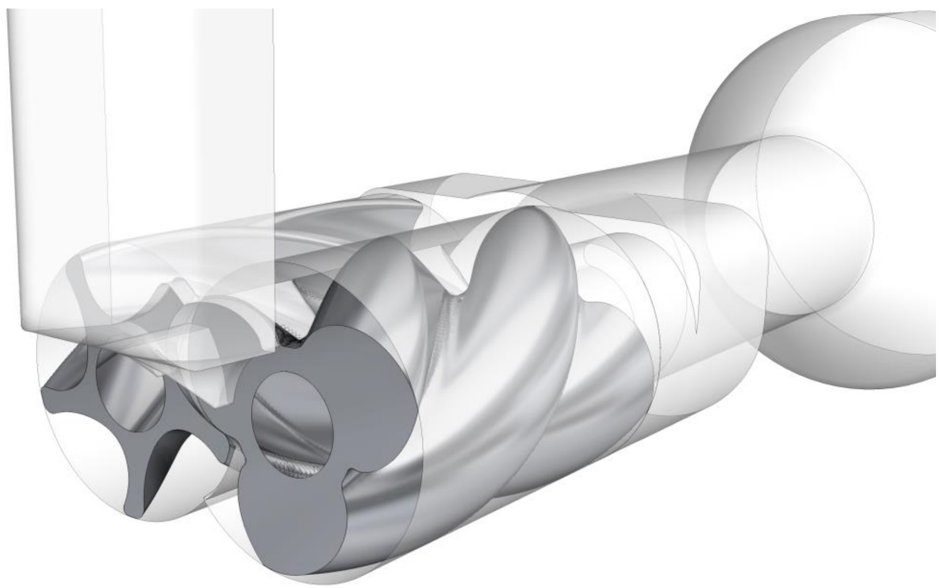
July 11 -14, 2016

PURDUE UNIVERSITY

Center for High Performance Buildings at Purdue

Introduction

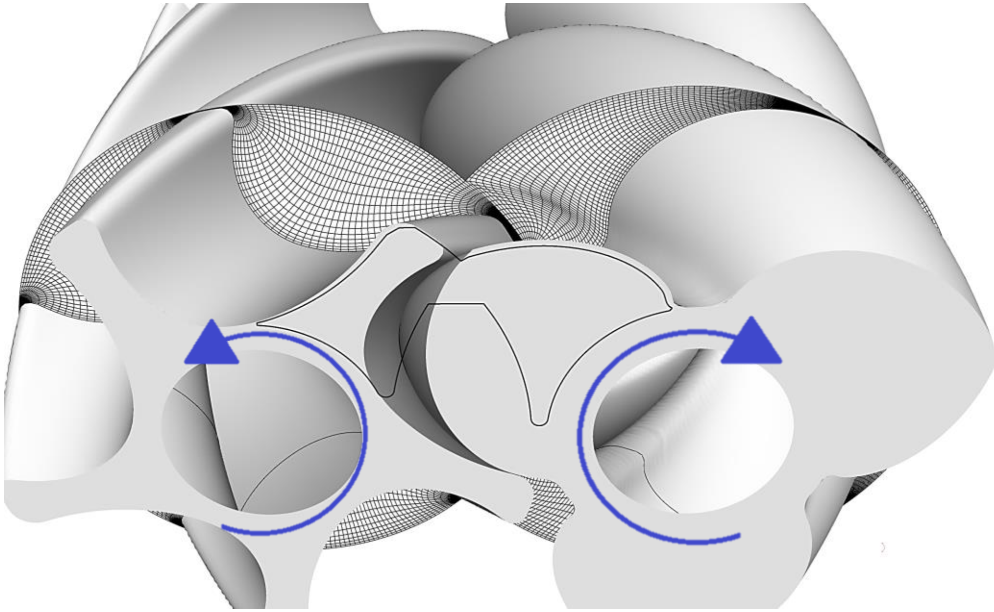
- Simulation of the twin screw expander SE-51.2 from TU Dortmund University including radial and axial clearances with commercial CFD solver ANSYS CFX
- Comparison of simulation results with measurements



Numerical model of the twin screw expander SE-51.2

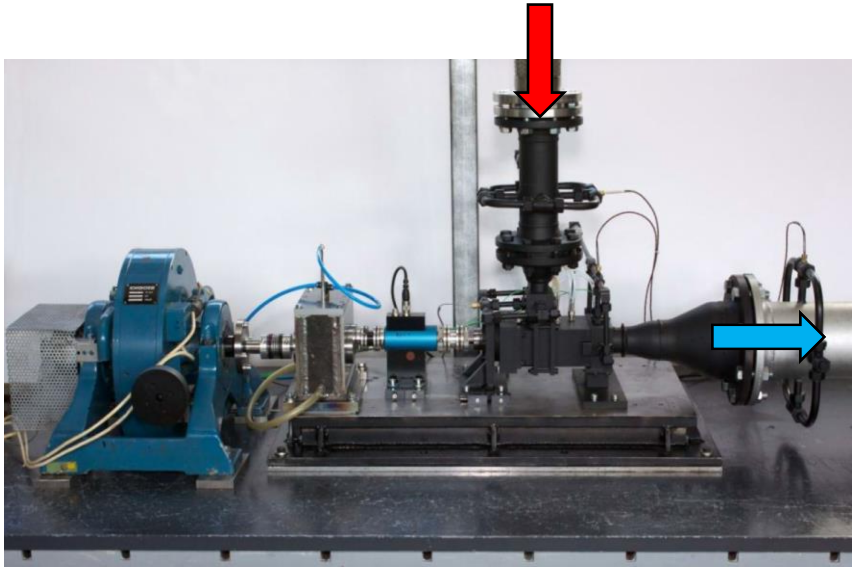
Introduction

- Challenges
  - » Time dependent change of complex rotor chamber volume
  - » Complex flow characteristics
  - » Compressible fluids
  - » Small clearances with high velocities



Numerical grid on a 2D cross-section through rotor chamber

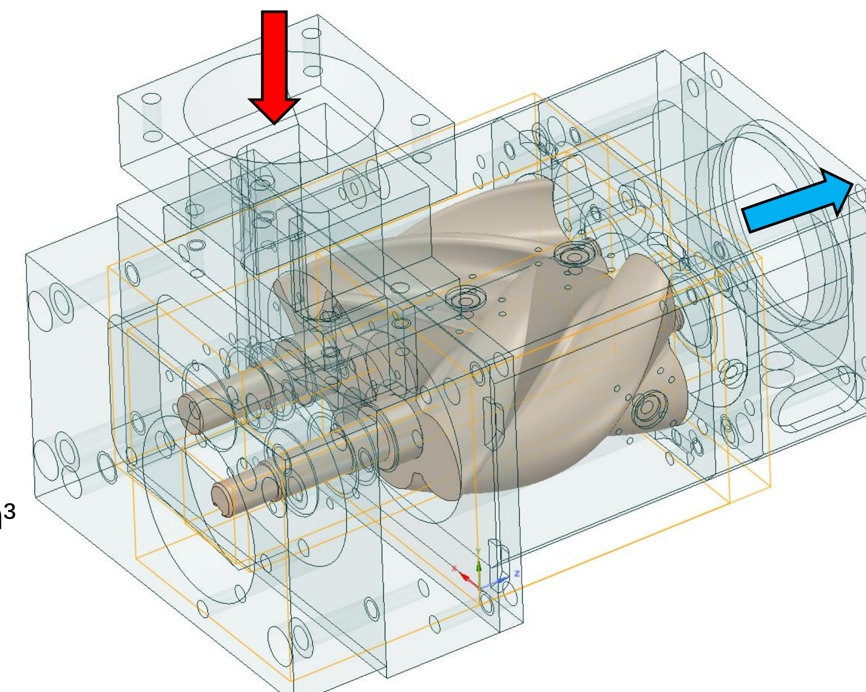
Geometry



Twin screw expander SE.51.2 and machine environment (Hütker, Brümmer, 2013)

- Experimental machine at Dortmund University
- Expanding high pressurized air
- Conversion of thermal into mechanical energy
- Can be used as an energy recovery system

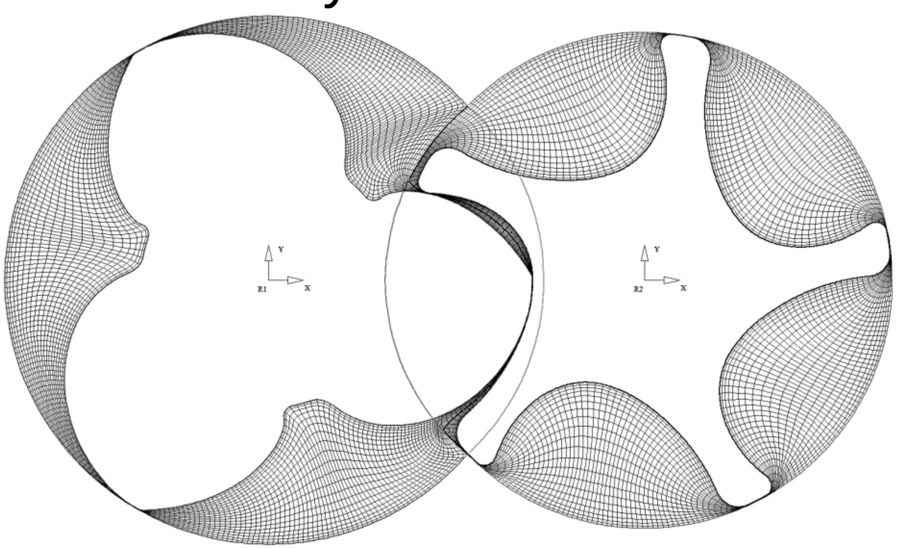
- Rotational speed (male rotor) up to 20 000 rpm
- Pressure ratios up to 6:1
- Built in volume ratio of 2.5
- Rotor center distance of 51 mm
- Displaced volume per male rotor revolution of 285 cm^3
- Runs dry and water-injected



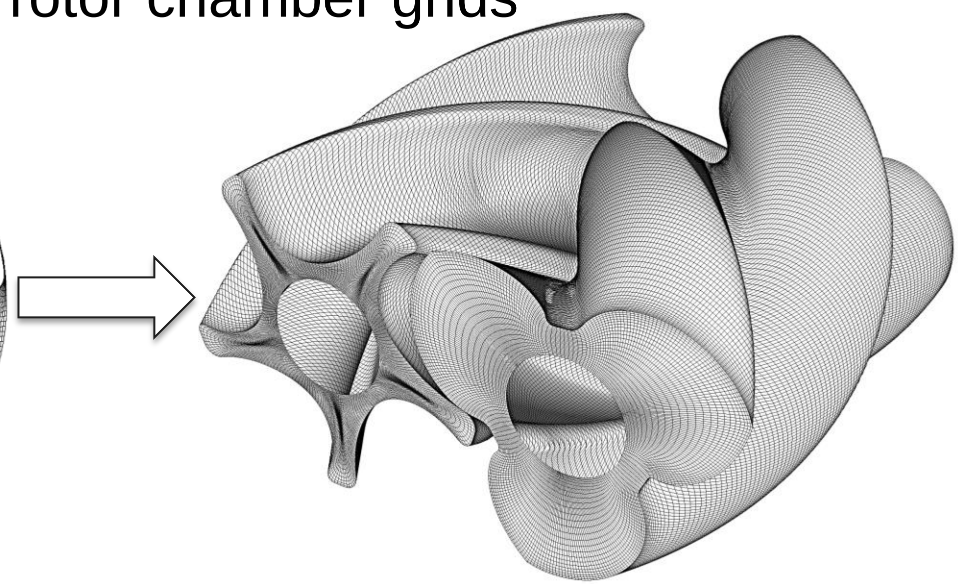
CAD model of the twin screw expander SE-51.2

Meshing

- Rotor chamber modeling with TwinMesh
  - » Generation of numerical grids for 2D cross-sections for each rotor position which has to be simulated
  - » 3D grids based on the 2D cross-sections
  - » Only hexahedral cells for rotor chamber grids

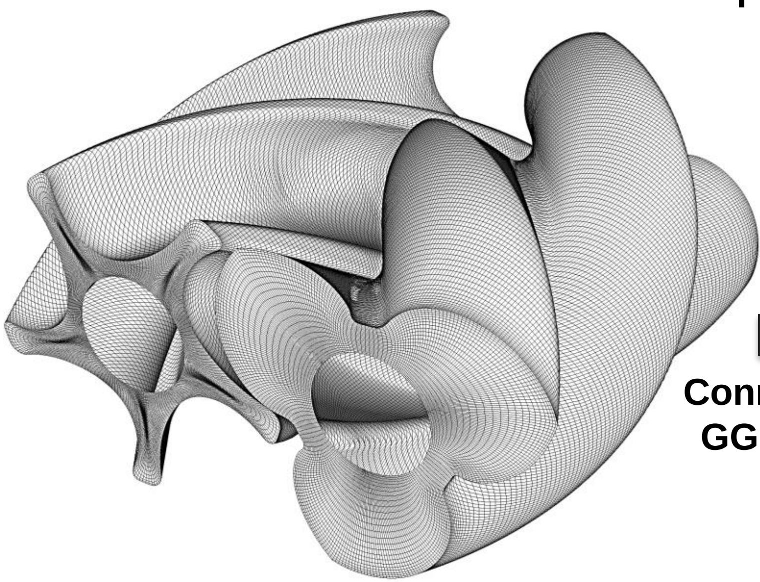


2D grid of the rotor chamber for a specific rotor position



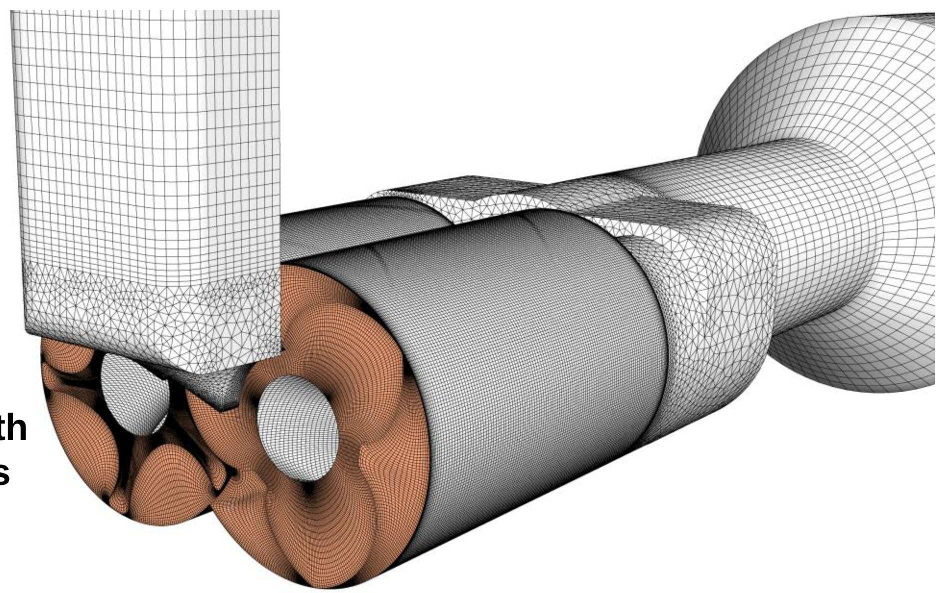
Meshing

- Assembly of rotary and stationary mesh parts
  - » Rotor meshes: approx. 1 million hexahedral cells per rotor (including gap meshes)
  - » Stator meshes: approx. 480 000 cells (hexahedrons, tetrahedrons and prisms)



3D grid of the rotor walls for specific rotor position

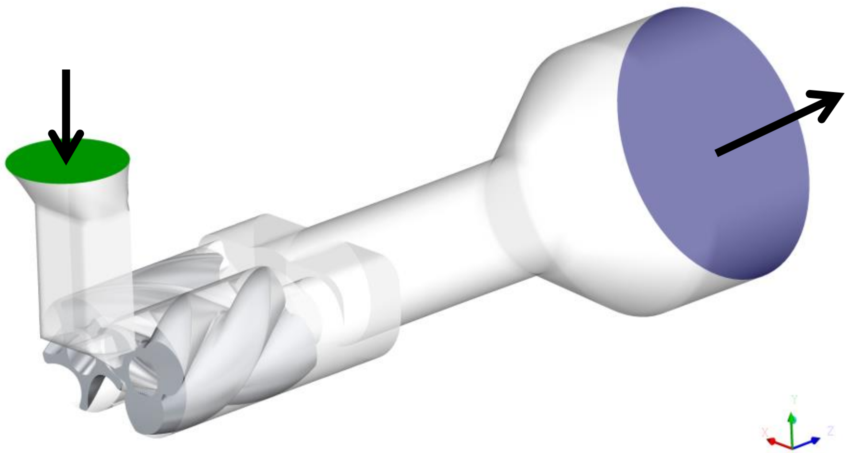
Connection with GGI interfaces



Assembly of rotor and stator meshes including front gaps

Simulation Setup

- Reference operating point (dry running)
  - » Pressure ratio inlet to outlet: 4 to 1, inflow at 90°C
  - » Revolution speed of 4000 rpm
  - » Fluid: air as ideal gas with SST turbulence model
  - » 1° angle step (male rotor) per simulated timestep
  - » Housing gap size: 80 µm
  - » Front gap size:
    100 µm (pressure side)
    170 µm (discharge side)
  - » Intermesh clearance:
    approx. 70 µm, variable in the real machine, because rotors are not synchronized



Simulation Setup

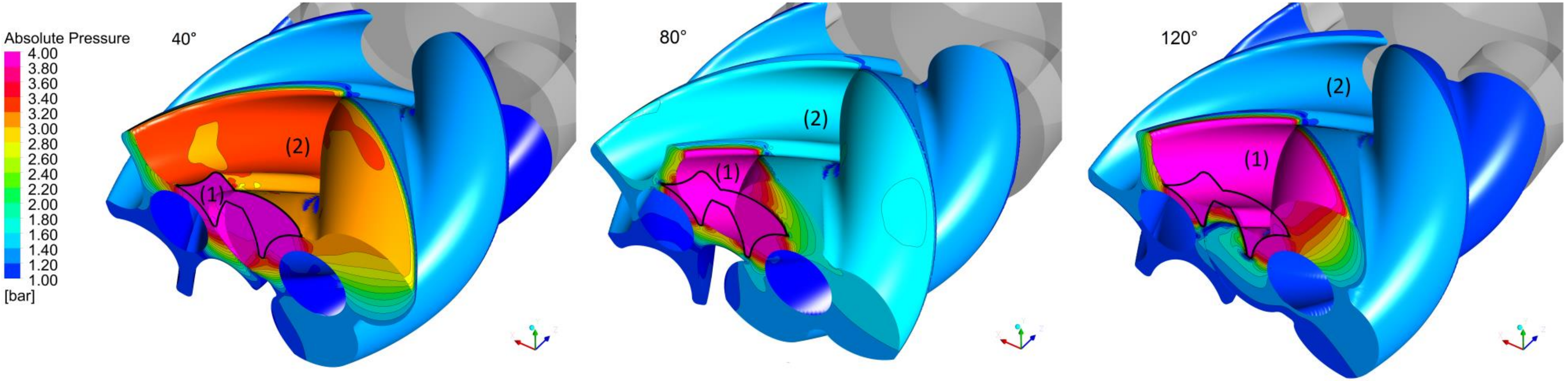
- Variations of the numerical model
  - » Rotation speed variation from 1 000 to 16 000 rpm
  - » Change of clearance size: +25% for housing and front gaps, approximately +50% for intermesh clearance and blow hole
  - » Decreasing the mesh resolution for the rotor chamber volume from approximately 1 million to 52 000 elements per rotor

Simulation Setup

- Clearances
  - » Variation of housing gaps is performed by a normal scaling of the rotors (both, male and female rotor)
  - » Due to the scaling, also intermesh clearance and the blow hole will increase
  - » Thus, all radial clearances (RC) in the numerical model are increased when scaling the rotors
  - » Variation of front gaps (FG) does not affect RC
  - » Clearance between stator/bearings and rotor shafts is neglected
    ➔ bearing leakage not included in the simulation

Simulation Results

- Pressure field on the rotors (4000 rpm)



Filling of chamber (1) started while air supply for chamber (2) was cut since lobes reached the control edges of the inflow port.

Once supply is cut, pressure in chamber (2) decreases due to expansion of its volume.

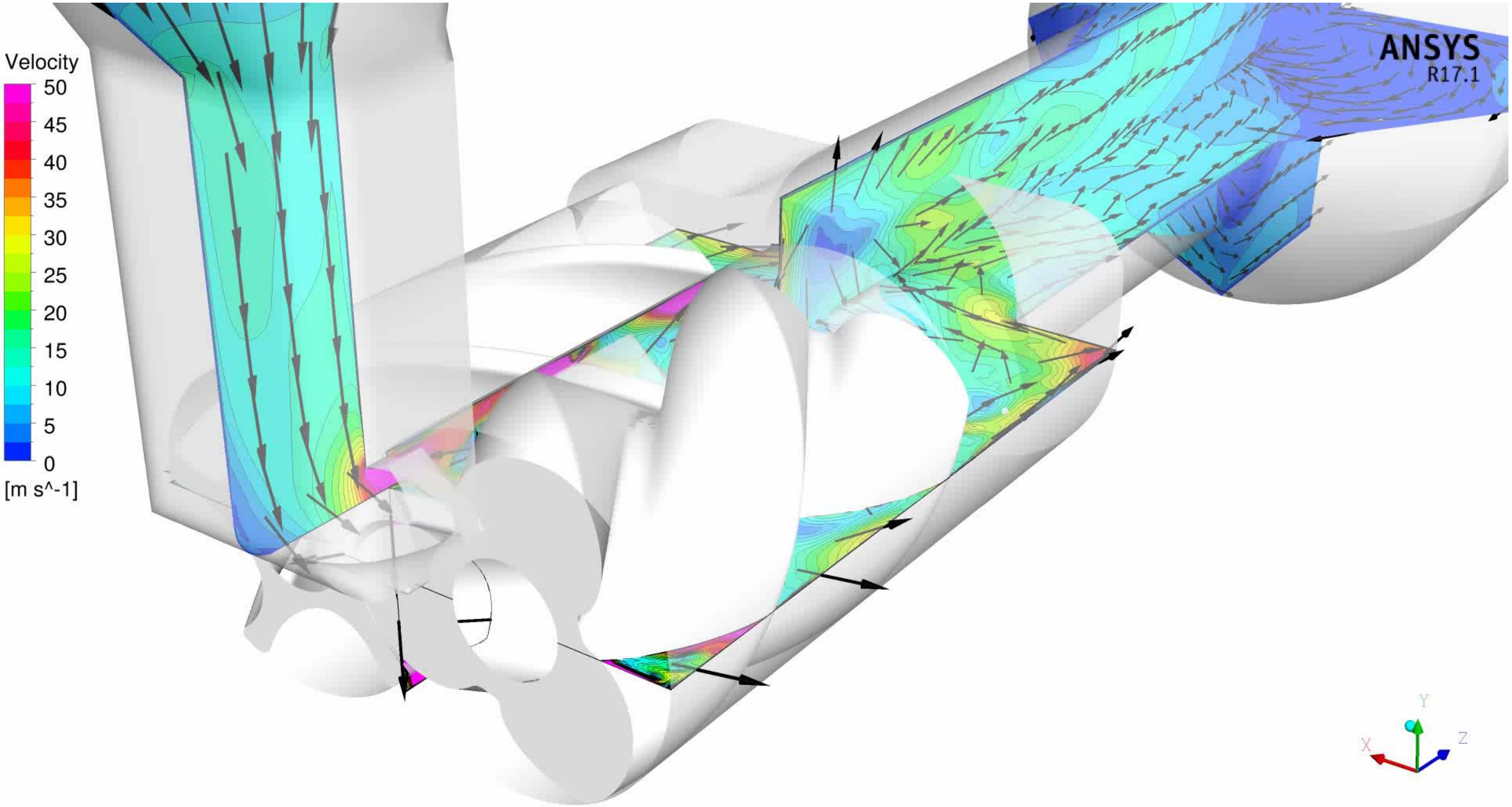
Ongoing inflow of air leads to nearly constant pressure level in chamber (1), although its volume is increasing.

Pressure in chamber (2) is further decreasing until it adapts to ambient pressure at the point of discharge.

Pressure level in chamber (1) will remain at its high level until the lobes reach the control edges again (see 40° position).

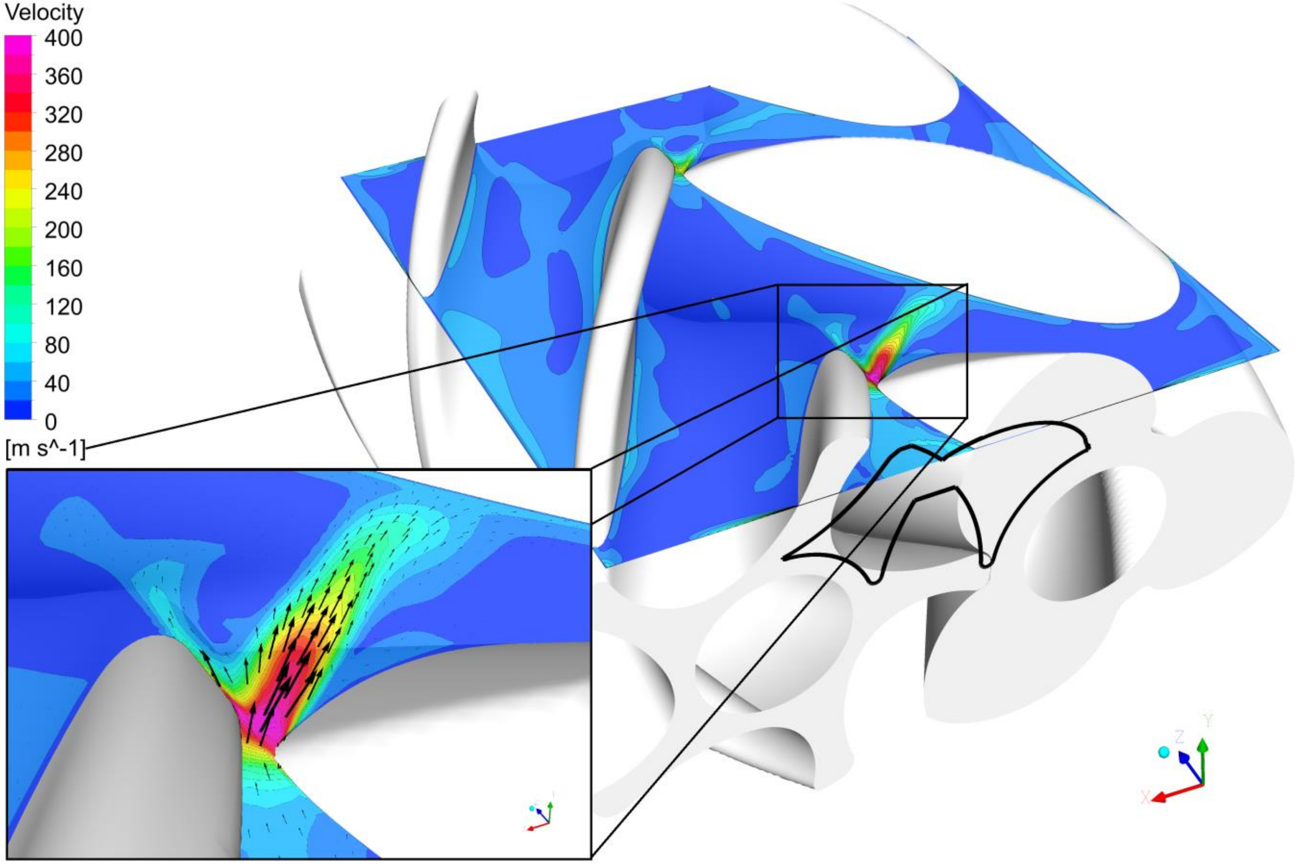
Simulation Results

- Velocity field on 2D cross-section planes (4000 rpm)



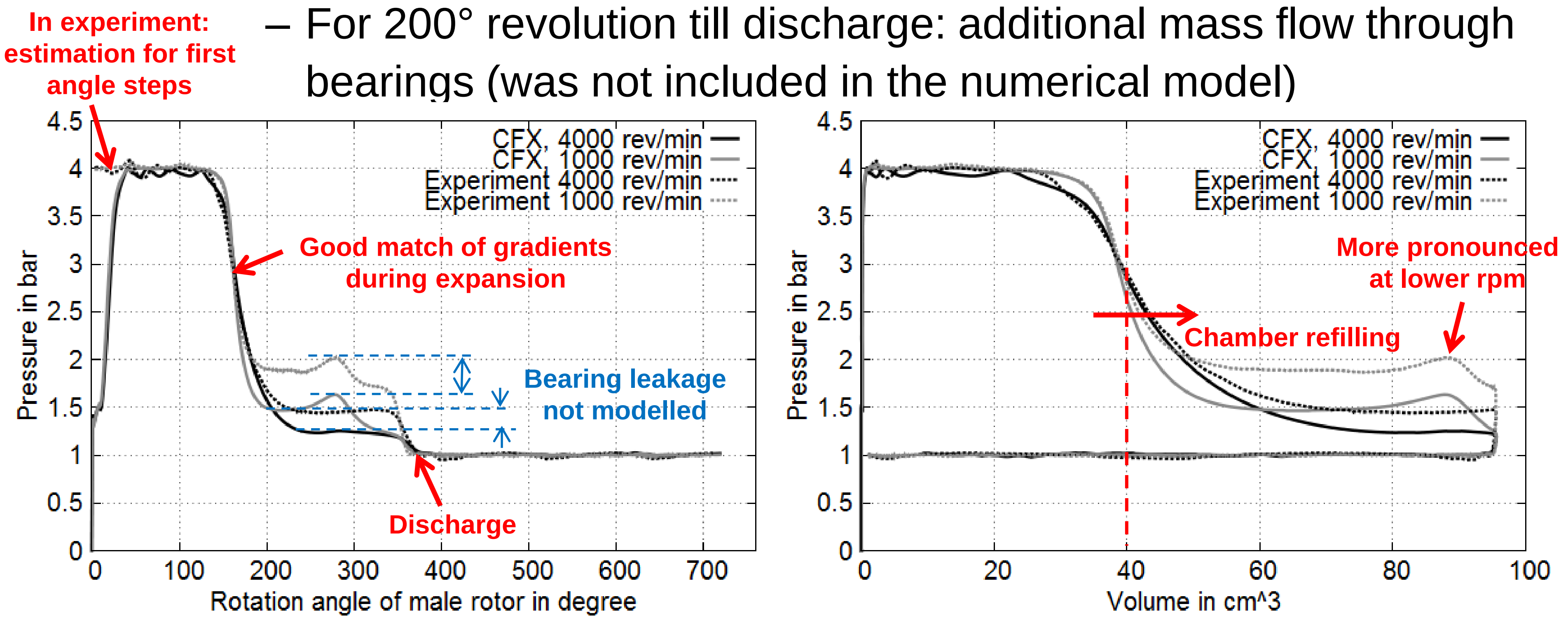
Simulation Results

- Velocity field on a 2D cross-section plane, including the leakage through the blow hole (4000 rpm)



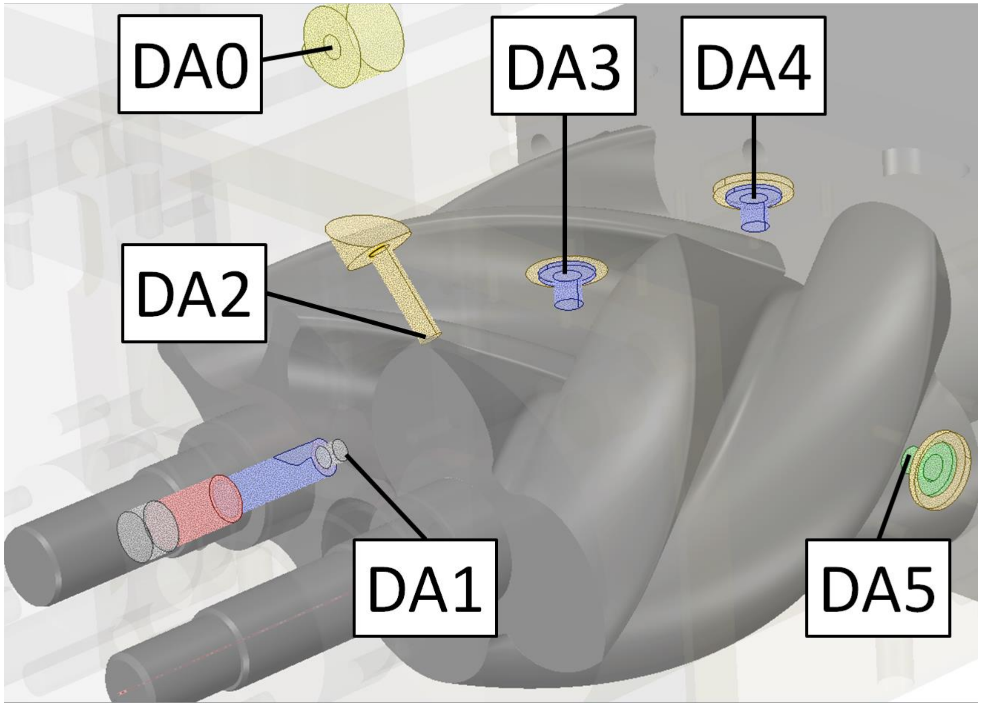
Simulation Results

- Pressure over rotation angle and p-V diagram
  - » Comparison with experiment for two rotational speeds:
    - – Very good overall qualitative agreement
    - – For 200° revolution till discharge: additional mass flow through bearings (was not included in the numerical model)



Simulation Results

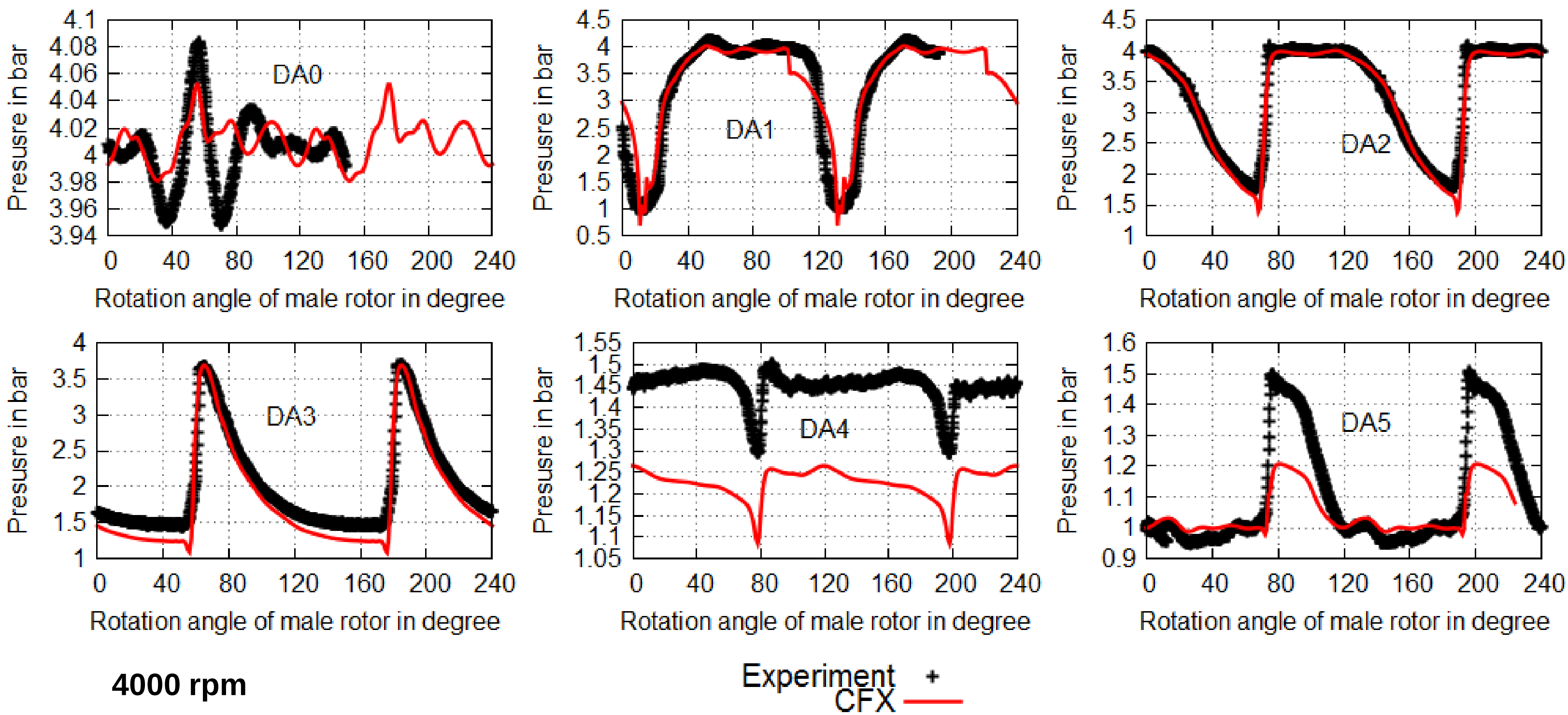
- Pressure comparison at measuring points “DA0” to “DA5”
  - DA0: inside inflow valve
  - DA1: axial measuring point near high pressure port
  - DA2 to DA5: radial measuring points from high to low pressure side



Location of pressure measurement points for the experimental setup and simulation

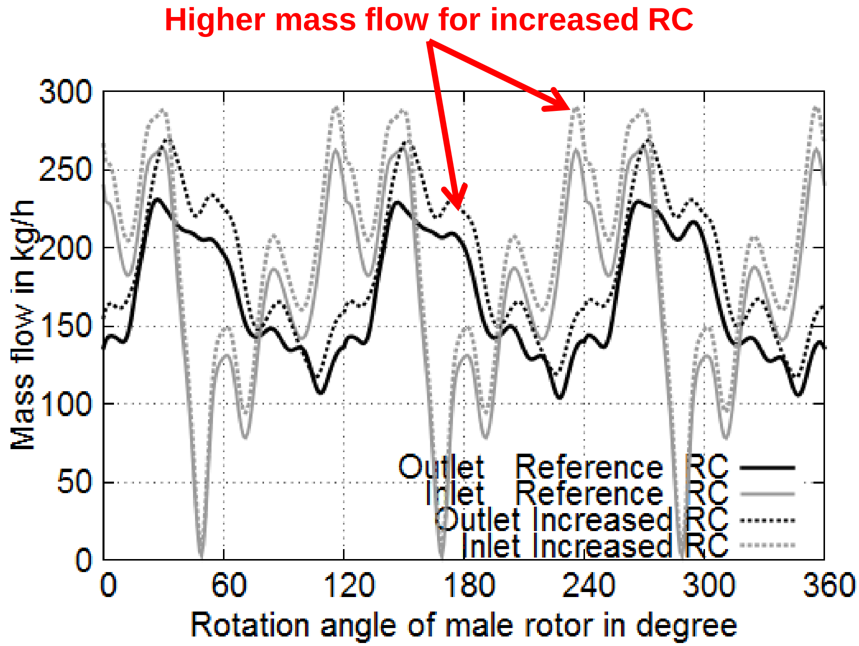
Simulation Results

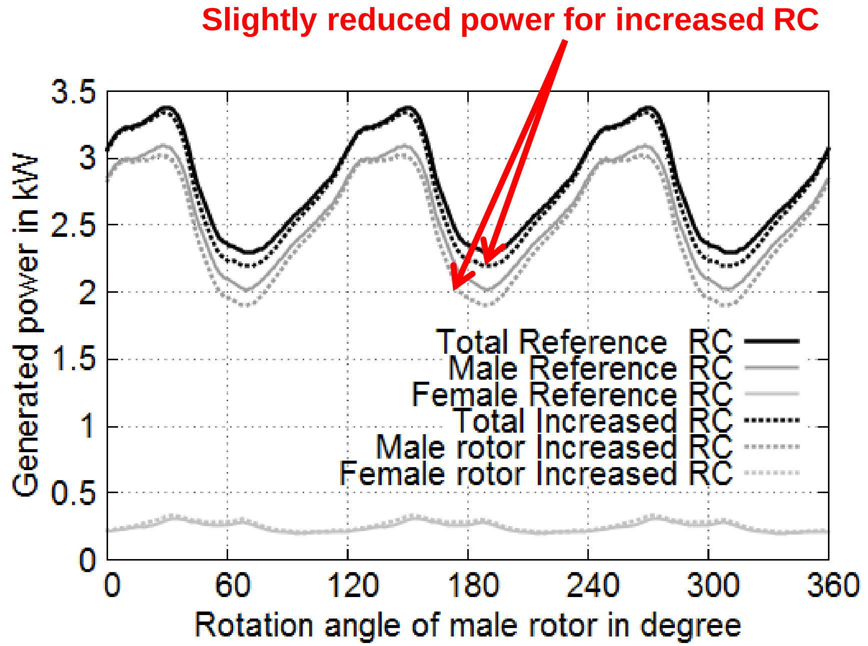
- Pressure comparison at measuring points DA0 to DA5
  - » Very good agreement, however influence of neglecting the bearing leakage noticeable at DA4 and DA5



Simulation Results

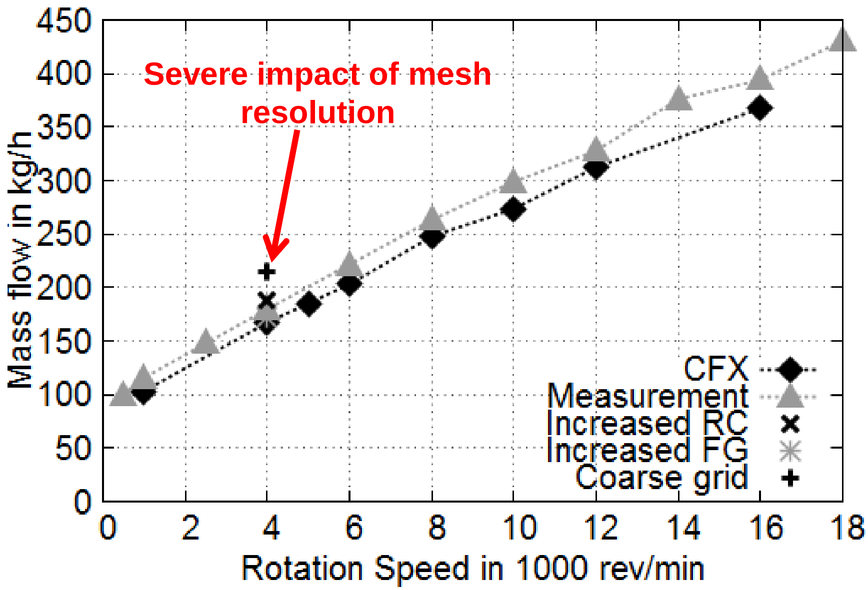
- Time-resolved mass flow and generated power over one male rotor revolution at 4000 rpm
  - » Comparison between reference and increased RC

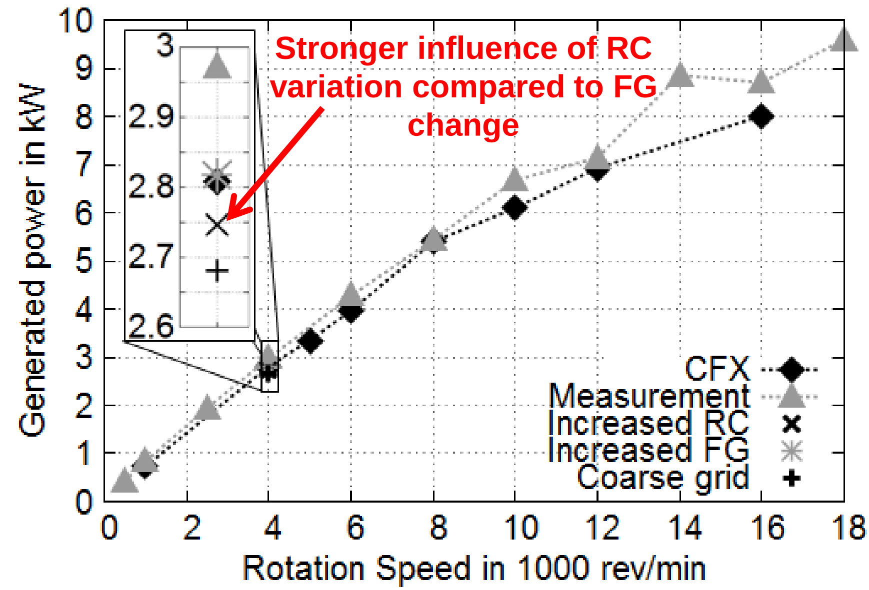




Simulation Results

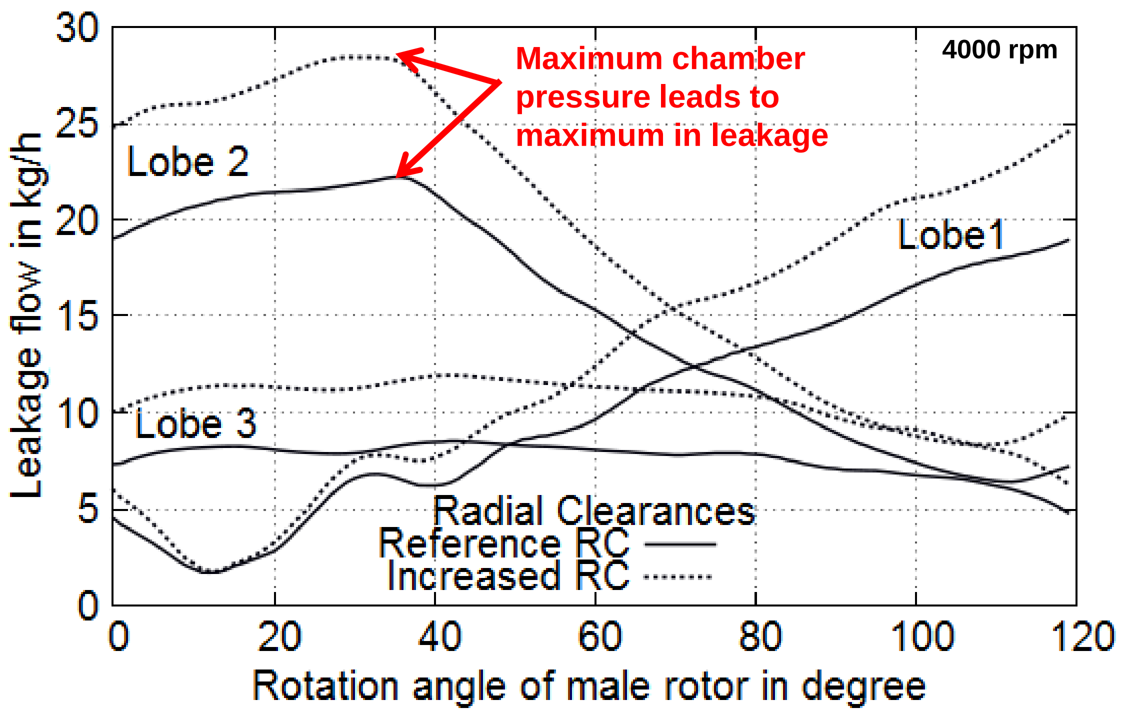
- Comparison of calculated mass flow and generated power for reference point and model variations
  - » Good agreement, slight underestimation in the simulation





Simulation Results

- Quantification of leakage mass flow through RC
  - » Reference RC (80 µm) and increased RC (100 µm)
  - » Possibility of sensitivity study concerning clearances sizes

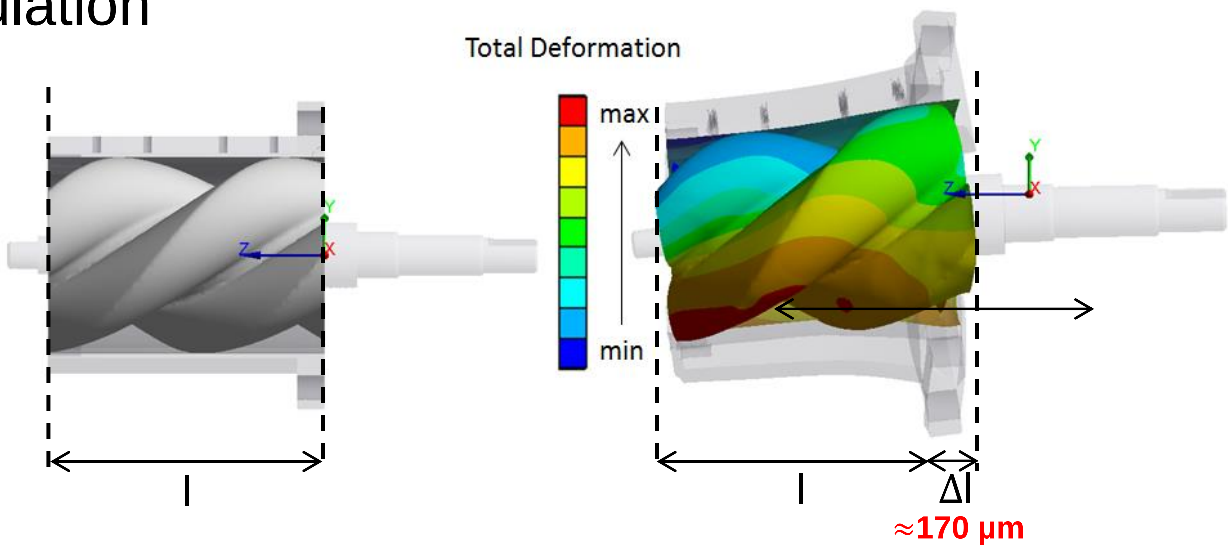


Summary

- Presentation of time- and space-resolved CFD results for a dry running screw expander simulated with ANSYS CFX
- Comparison of CFD results with measurements
- Sensitivity study regarding clearances and mesh resolution
- Chamber modeling with TwinMesh
- Neglection of an additional leakage source (bearing leakage) shows severe impact when comparing CFD to experiment

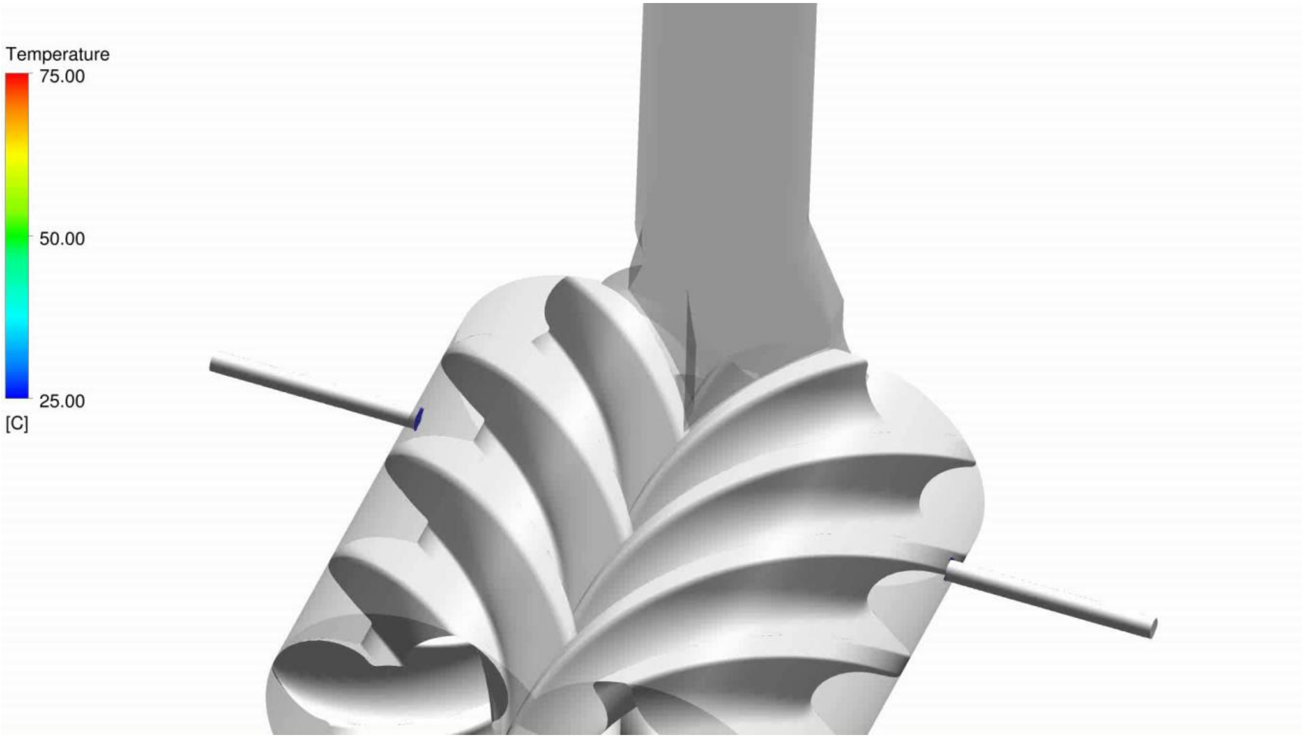
Outlook

- Fluid structure interaction (FSI)
  - » One way-coupling of fluid and structure for a specific point in time
  - » Calculating stator and rotor deformations based on the temperature and pressure loads derived from the CFD simulation



Outlook

- Injected screw machines
  - » Example of an oil-injected screw compressor



Outlook

- Extension of the present numerical model in order to capture all leakages (i.e. adding bearing leakage)
- Performing a separate quantification for the individual leakage flows (housing gaps, front gaps, intermesh clearance, and blow hole)
- Validating results for clearance mass flows and clearance variation with experimental data
- Validation of FSI results and multiphase (injected) cases with measurements

Thank you for your attention!

rainer.andres@cfx-berlin.de