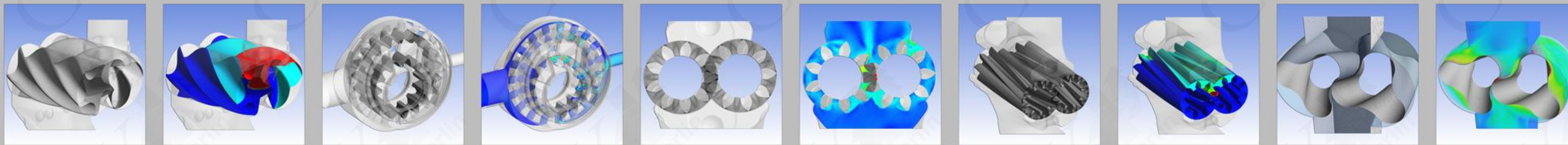


TwinMesh for Positive Displacement Machines: Structured Meshes and reliable CFD Simulations

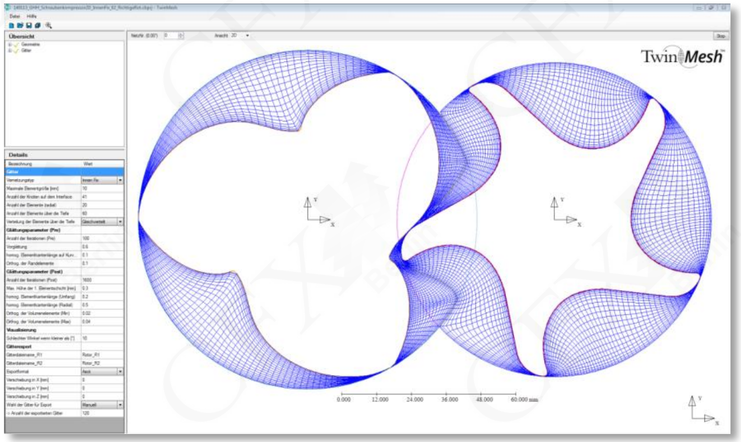
05.06.2014

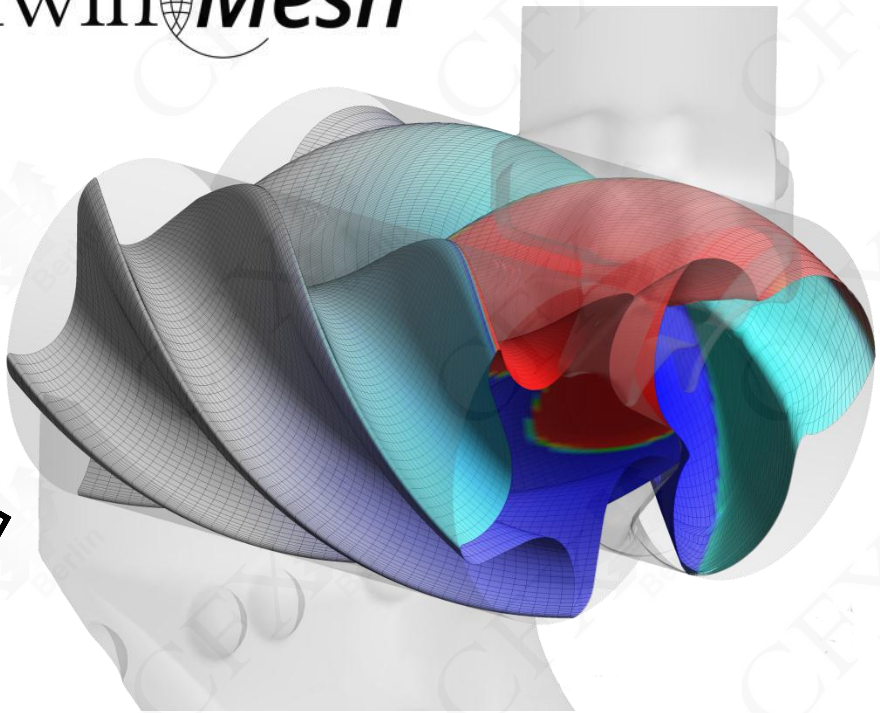
Dipl.-Ing. Jan Hesse, Dr. Andreas Spille-Kohoff
CFX Berlin Software GmbH
Karl-Marx-Allee 90 A
10243 Berlin

Contents

- Introduction
- Methods for chamber modelling
- TwinMesh
- Simulation results
- Summary

TwinMesh™

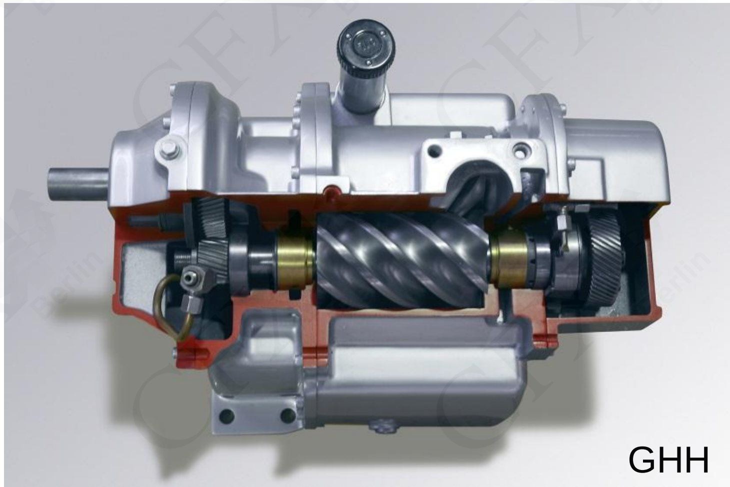




Introduction

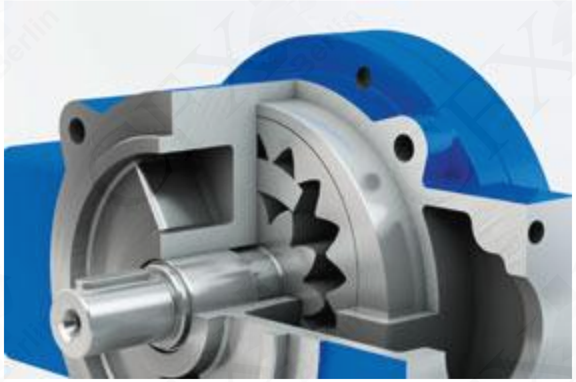
- Positive displacement machines (examples)

Screw compressor



Lobe pumps

Gear pumps



Introduction

- Characteristics of the geometries
  - Two rotating rotors (often screwed)
  - Size-changing chambers with very small clearances between the lobes and between rotors and casing

- Characteristics of the flow
  - Cavitation (Multiphase)
  - Non-newtonian fluid

  - Compressibility
  - Real-gas properties

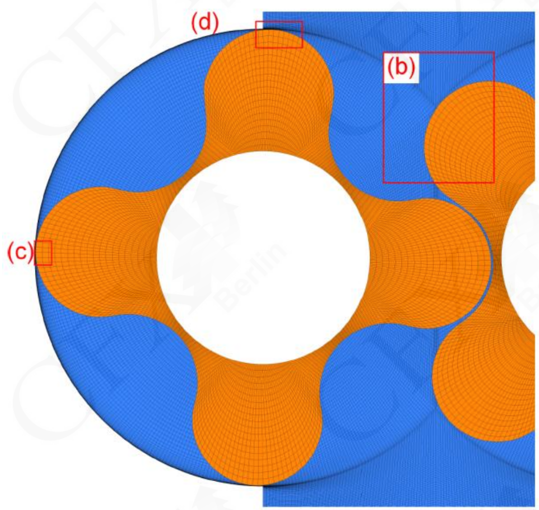
  - Turbulence
  - Viscous heating, etc.

Lobe pump

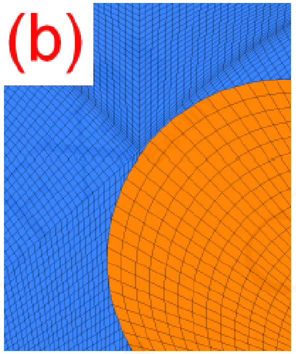
How can I model this complex behavior in my CFD-simulation?

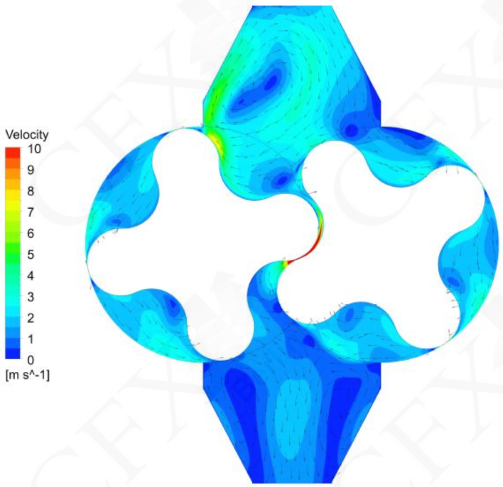
Methods for chamber modelling
Immersed-Solid

- Advantages
  - Fast – only one mesh for fluid and solid volumes
  - Variable time step control



- Disadvantages
  - Insufficient wall treatment
  - Multiphase is not available (e. g. Cavitation)
  - **Only incompressible fluids**
  - Possible numerical instabilities depending on local pressure gradients (e. g. gap flow), small time steps necessary
  - Very large number of elements depending on geometry especially for unstructured meshes



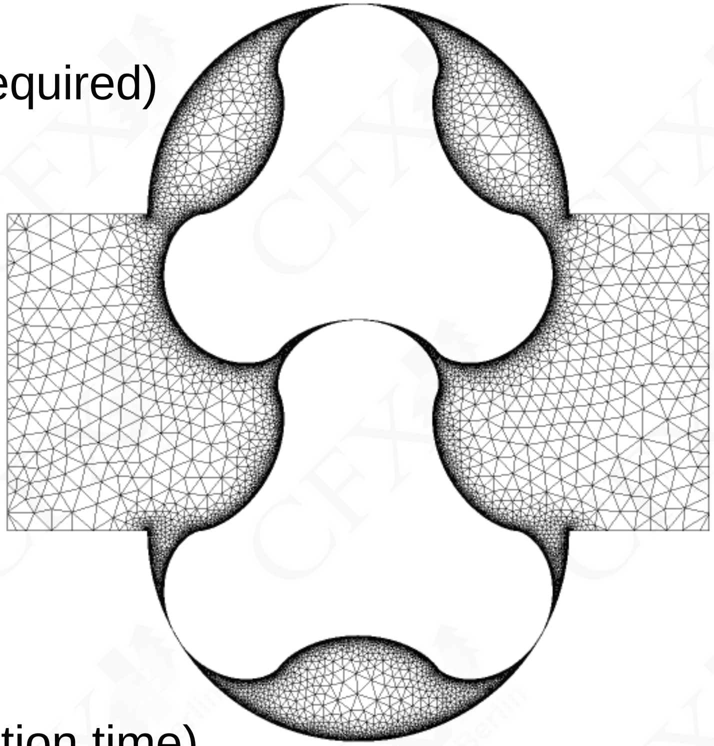


(c) $\varphi = 15°$

Schwotzer, T.: „Simulation einer Drehkolbenpumpe mit der Immersed-Solid-Methode“, Bachelorarbeit, Technische Universität Berlin, 2009

Methods for chamber modelling
Mesh Deformation and Remeshing

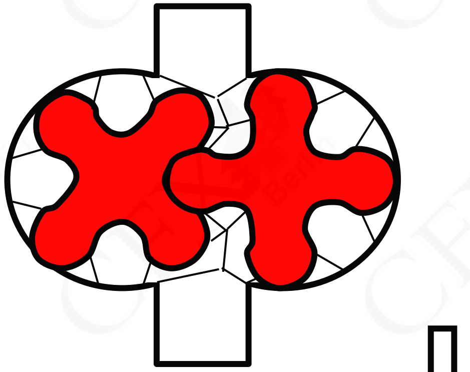
- Advantages
  – Automatic mesh generation (less manpower required)
  – Fluid volume is represented by the mesh
  – Full model support
    (e.g. Multiphase with Cavitation,
    turbulence model)

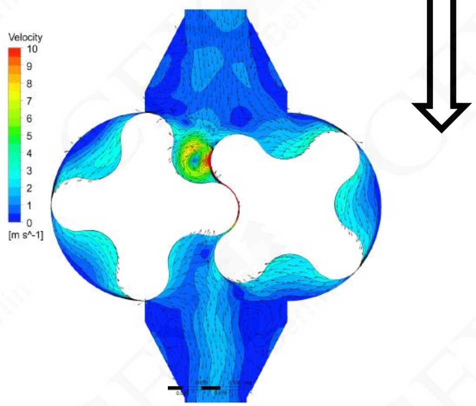


- Disadvantages
  – Mesh-generation for almost each iteration
    (increasing computation time)
  – Mesh generation leads to very high
    element numbers in gaps (increasing computation time)
  – Mesh quality issues due to mesh deformation and
    element topology when using remeshing
  – Numerical errors due to frequent interpolation of calculation results
    between different meshes

Methods for chamber modelling
Manual Generation of Structured Hexahedral Meshes

- Idea
  - Manual grid generation in ANSYS ICEM CFD Hexa for many rotor positions per rotation
- Advantages
  - Best mesh and numerical quality
  - High resolution of gaps is possible
  - Element topology allows manageable model size
  - No interpolation errors since the grid topology remains the same
- Disadvantages
  - Extremely **high manual effort**: grid generation for 2D-models would need 4 weeks



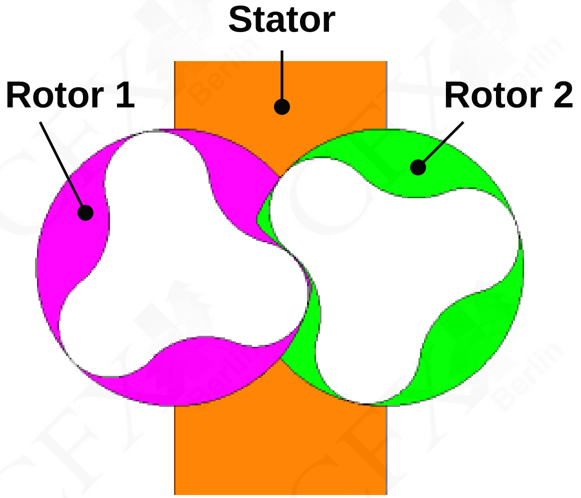


Fuchs, M.: „Numerische Simulation der instationären Strömung in einer Drehkolbenpumpe“, Bachelorarbeit, Technische Universität Berlin, 2010

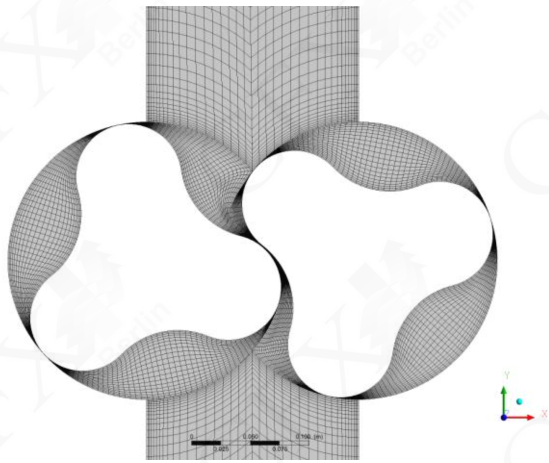
TwinMesh
Seven steps from CAD to Mesh

- TwinMesh is a novel software, developed by CFX Berlin Software GmbH which generates high-quality hexahedral meshes for the rotating parts of axis parallel rotary positive displacement machines.

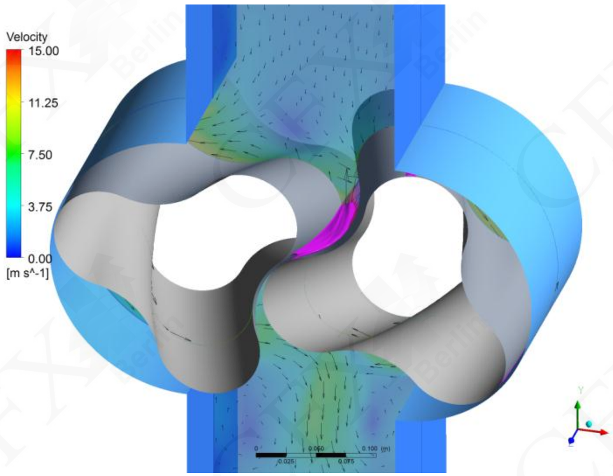
Simulation Workflow



Split of the simulation domains into steady and rotating parts

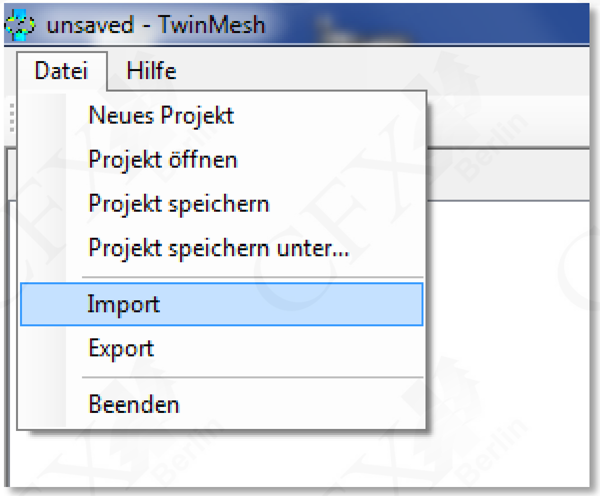


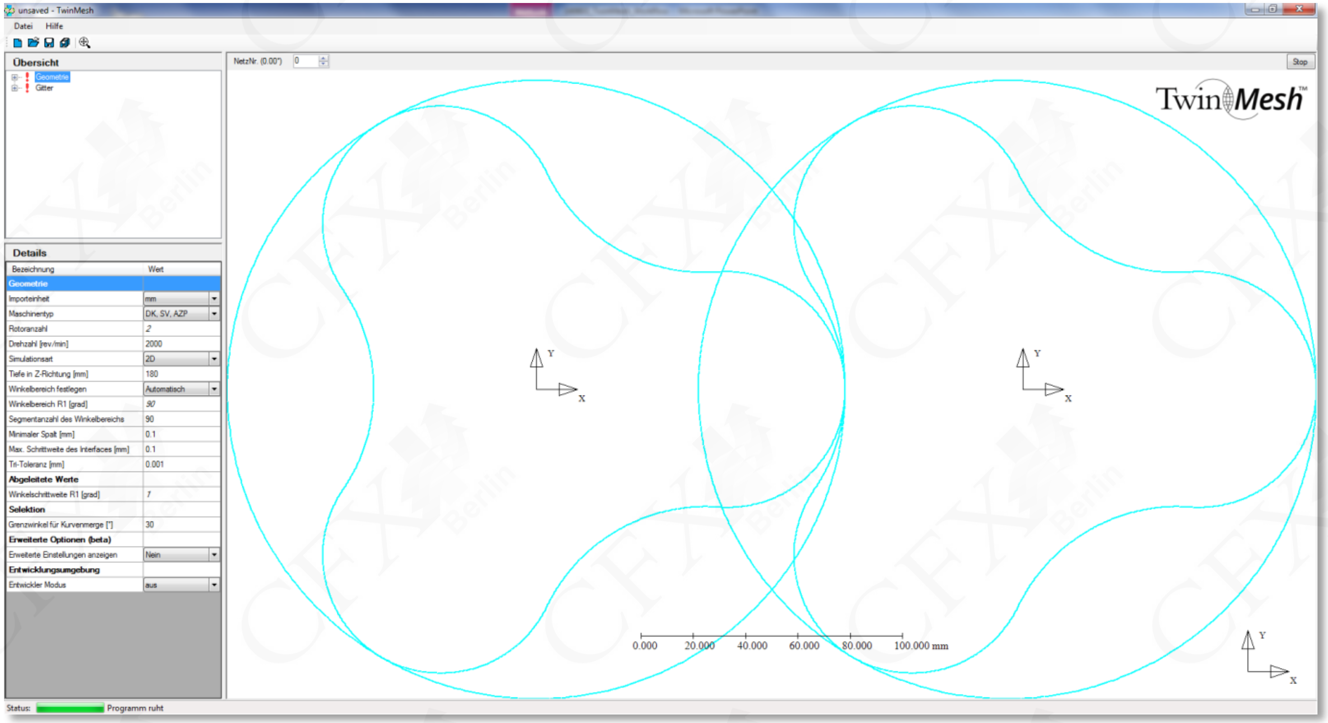
Grid generation for the rotors with **TwinMesh** and for the Steady parts with ANSYS ICEM CFD or ANSYS Meshing



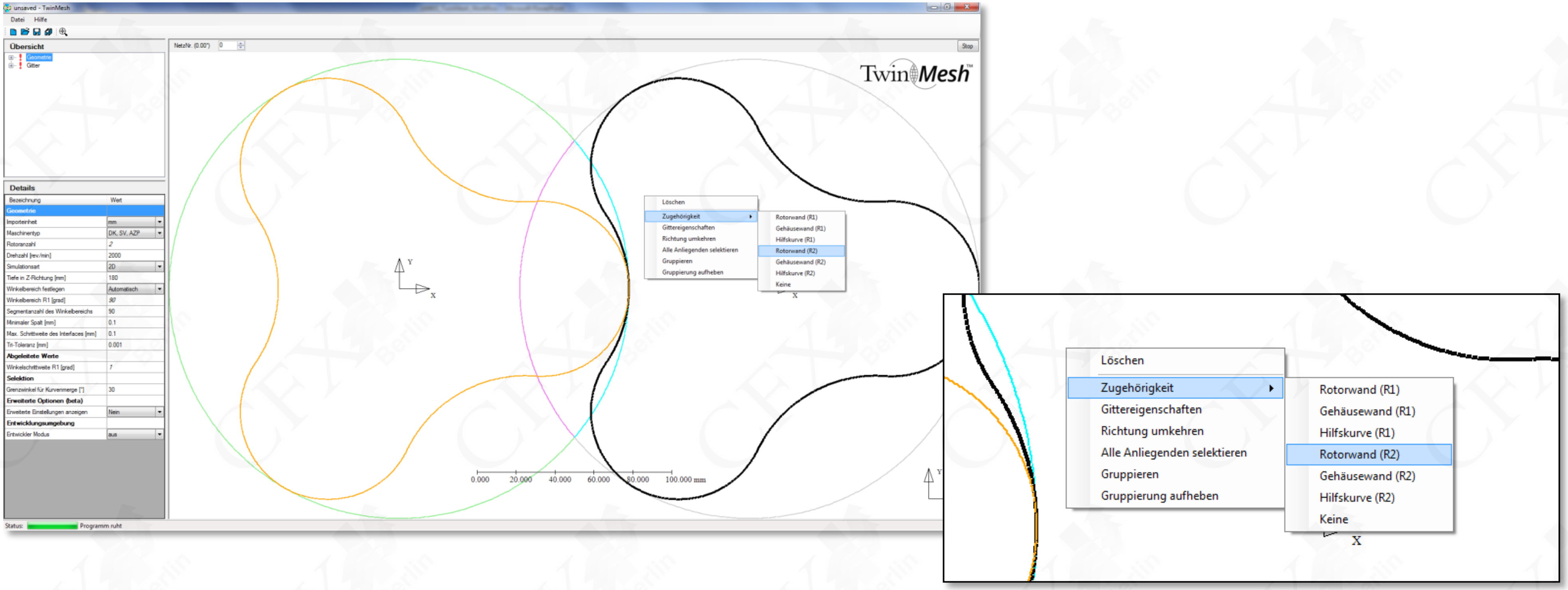
Numerical calculation with ANSYS CFX

- File format (2D cross section of the machine)
  - IGES
  - CSV-File with point coordinates

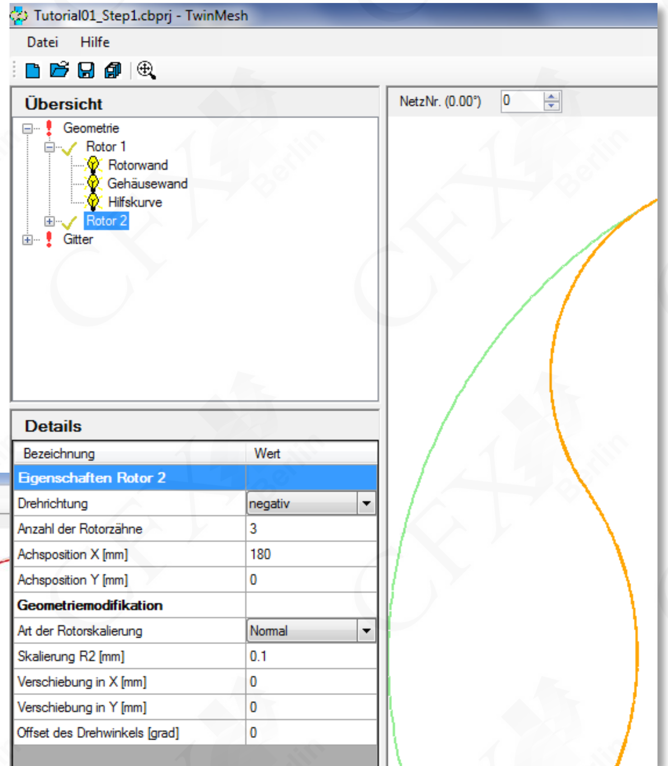


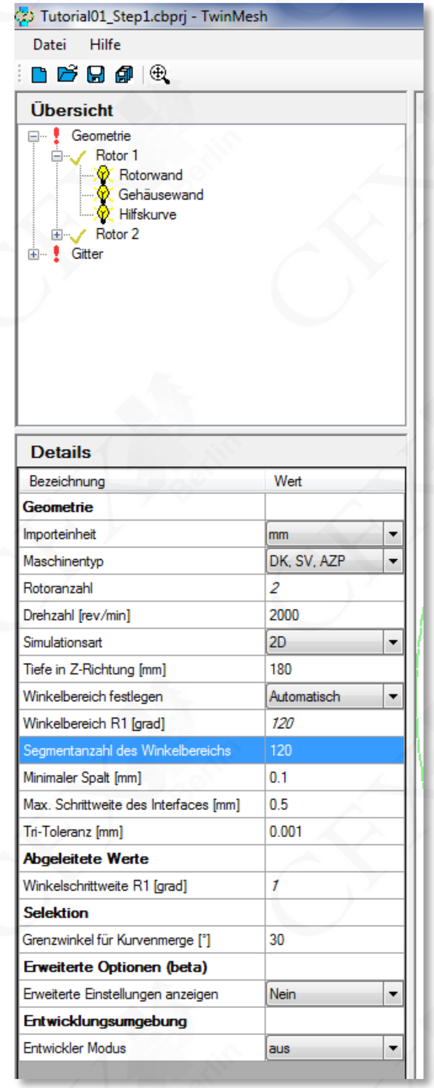


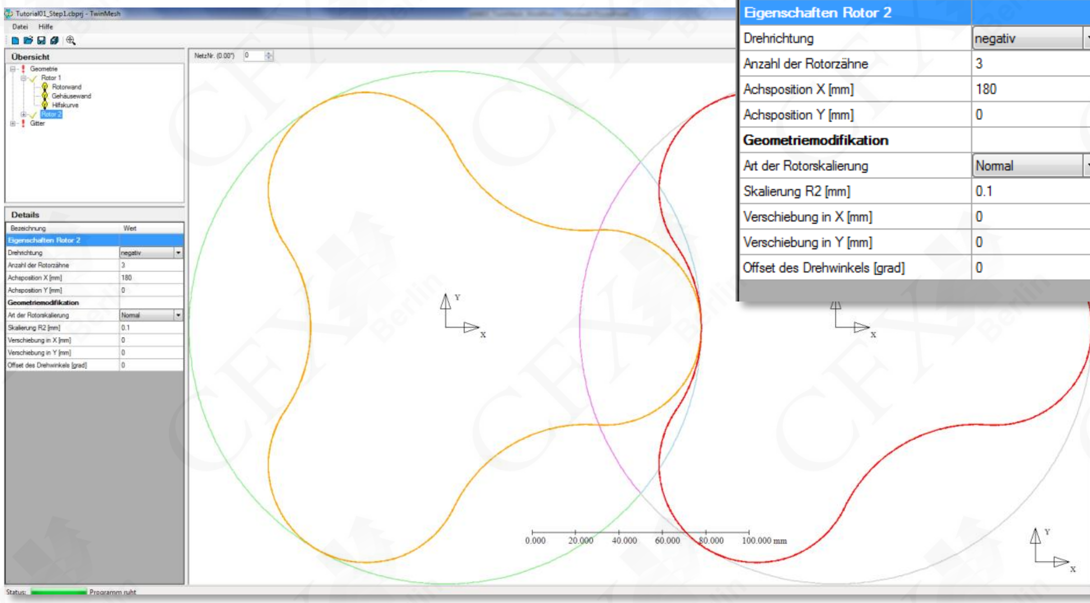
- Boundary types
  - Rotor curvature
  - Casing curvature
  - Additional curvature for interface creation



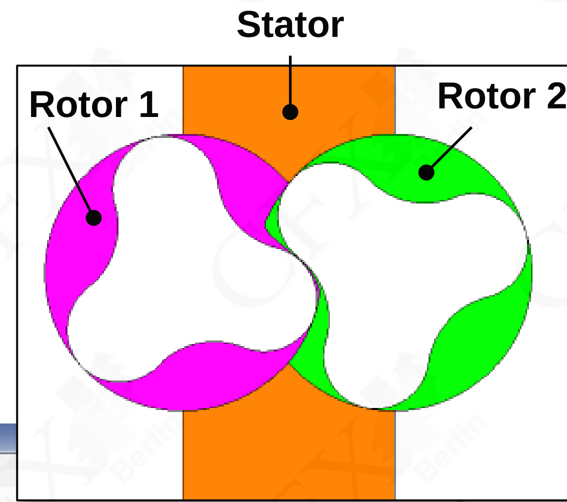
- Options
  - Number of lobes
  - Rotorposition
  - Rotor curvature modifications
  - Screwing
  - Angle per timestep
  - …

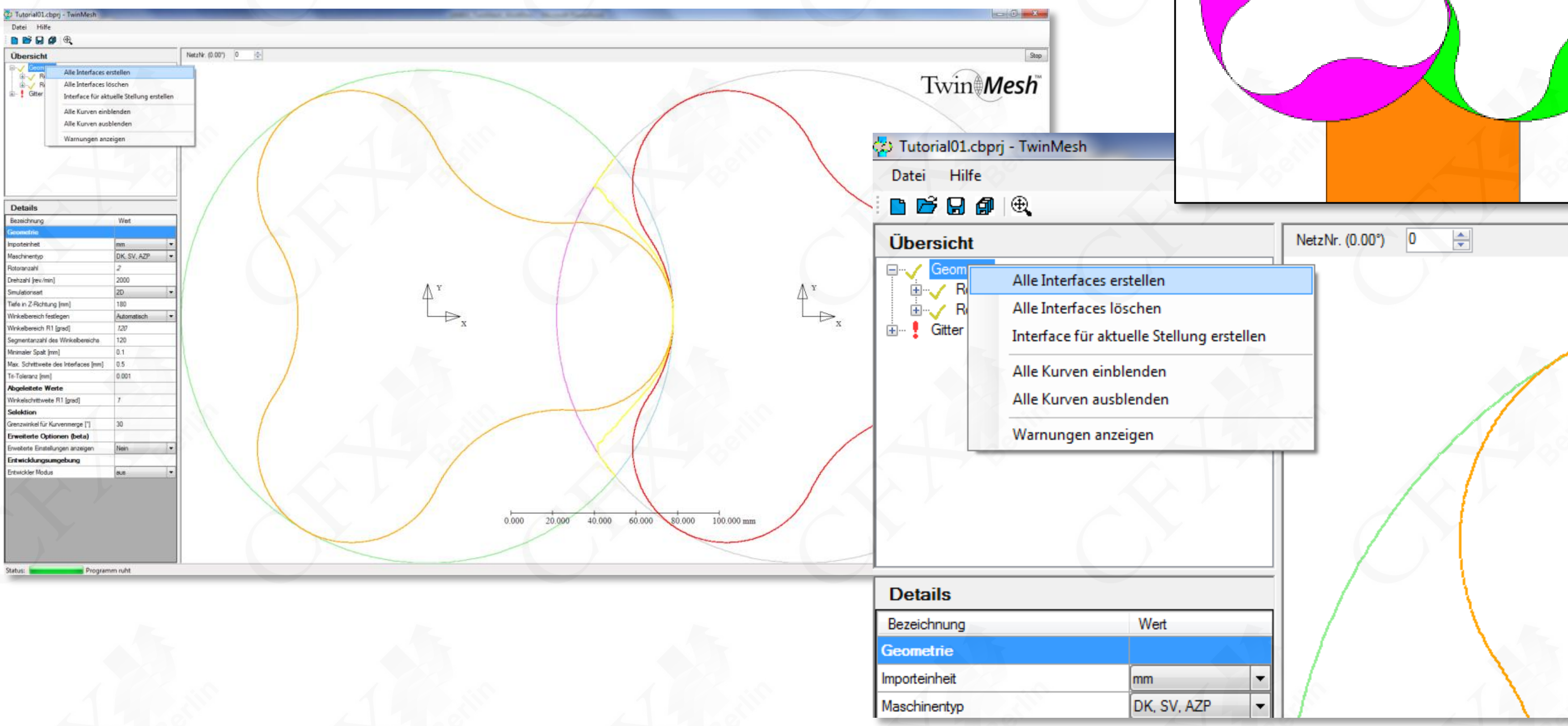




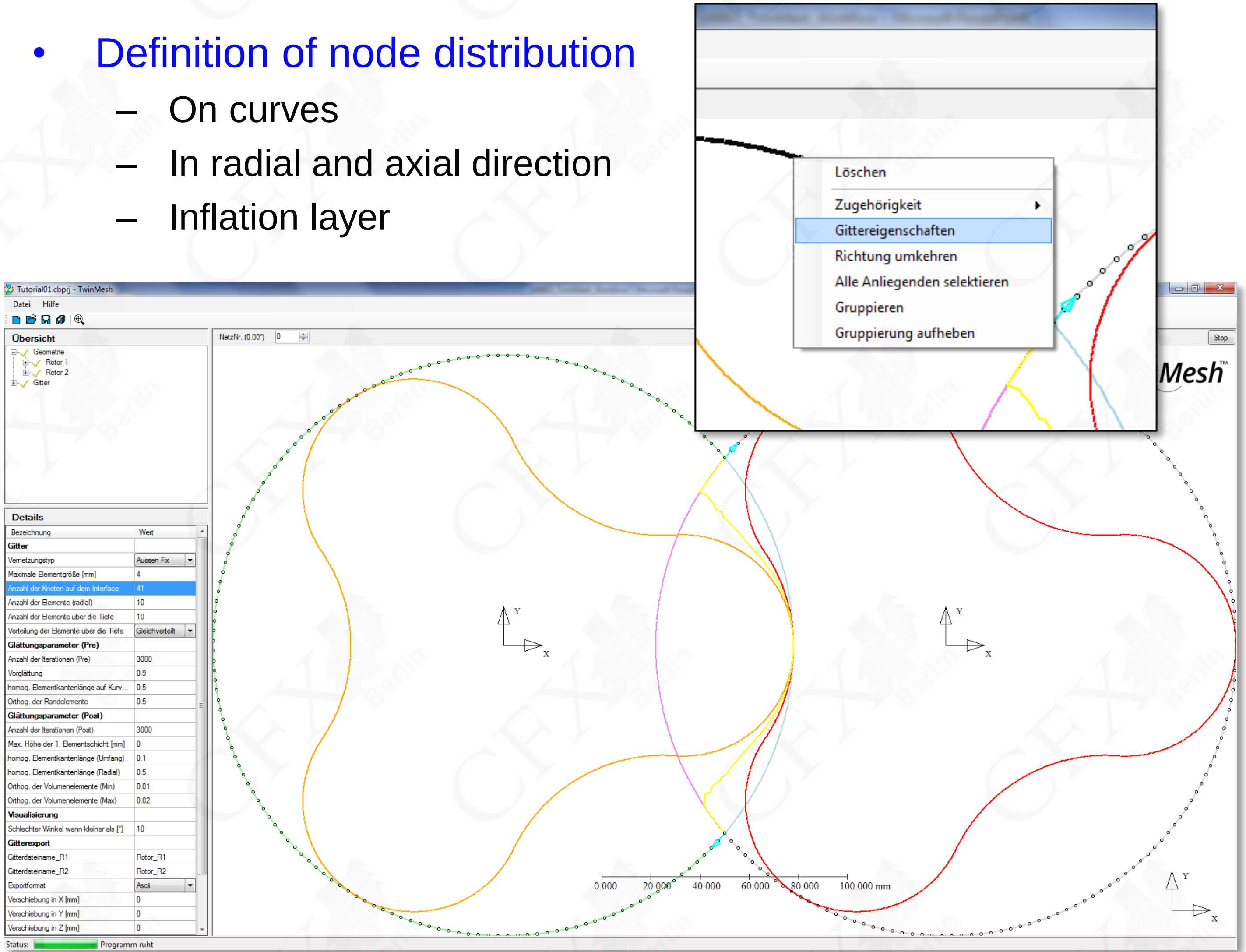


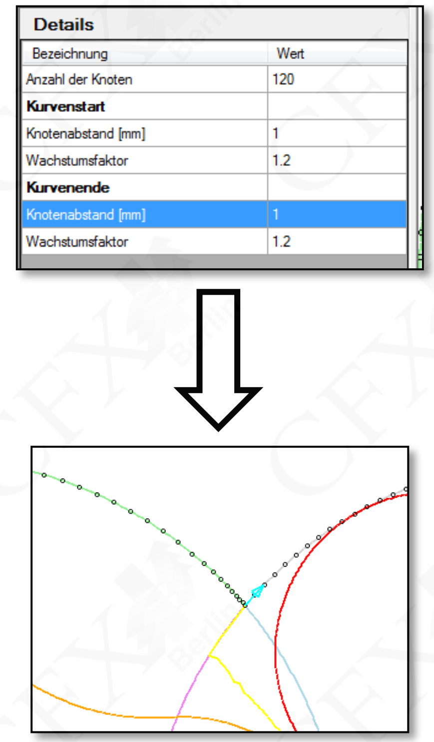
- Automatic contact definition for the rotor meshes
  - Interfaces between the rotor meshes are automatically generated depending on rotor curvature



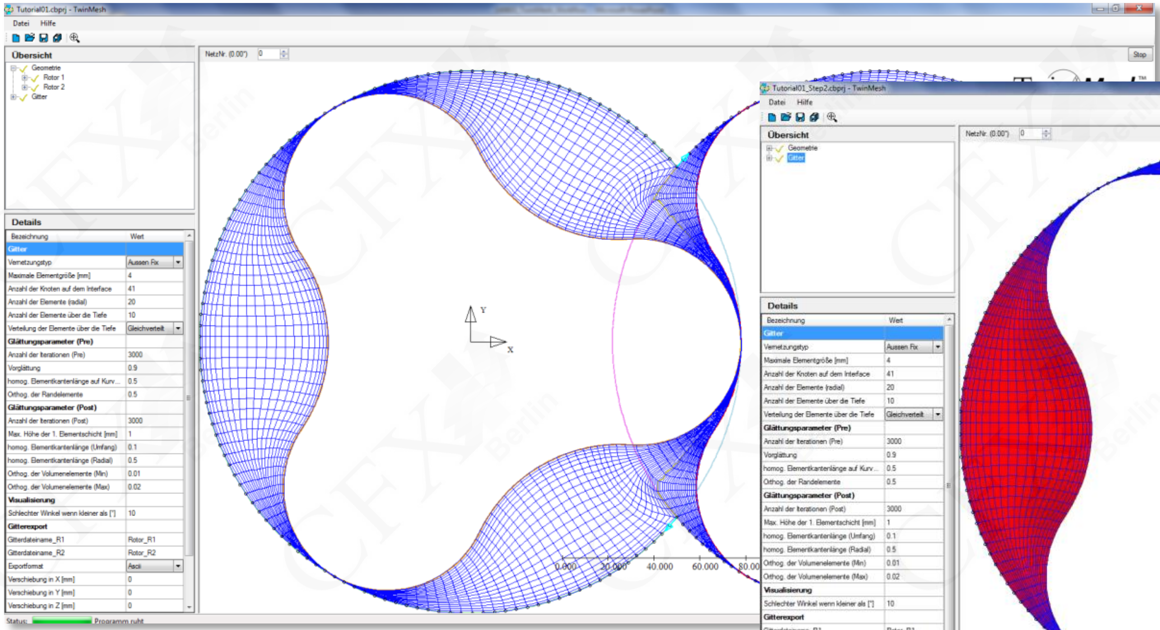


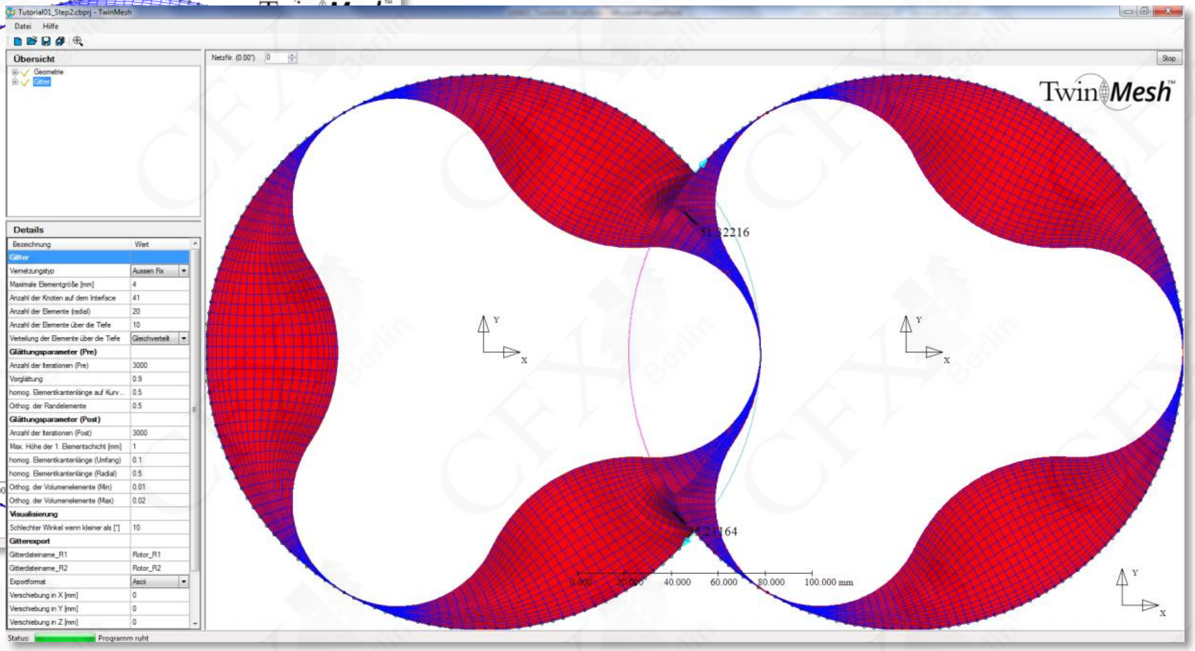
- Definition of node distribution
  - On curves
  - In radial and axial direction
  - Inflation layer



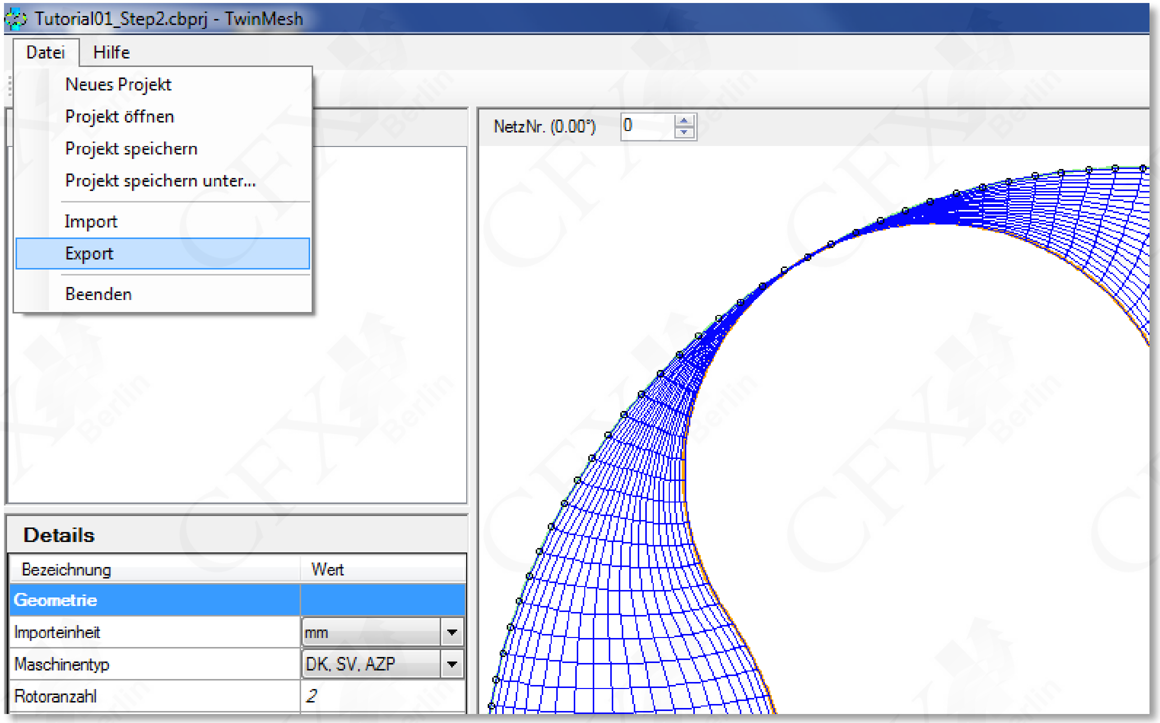


- Automatic mesh generation for each rotation angle
  - Smoothing algorithm depending on orthogonality and volume change
  - Different methods of mesh connection available (non-conforming or 1to1)
  - Visual and quantitative quality check tools available


Mesh

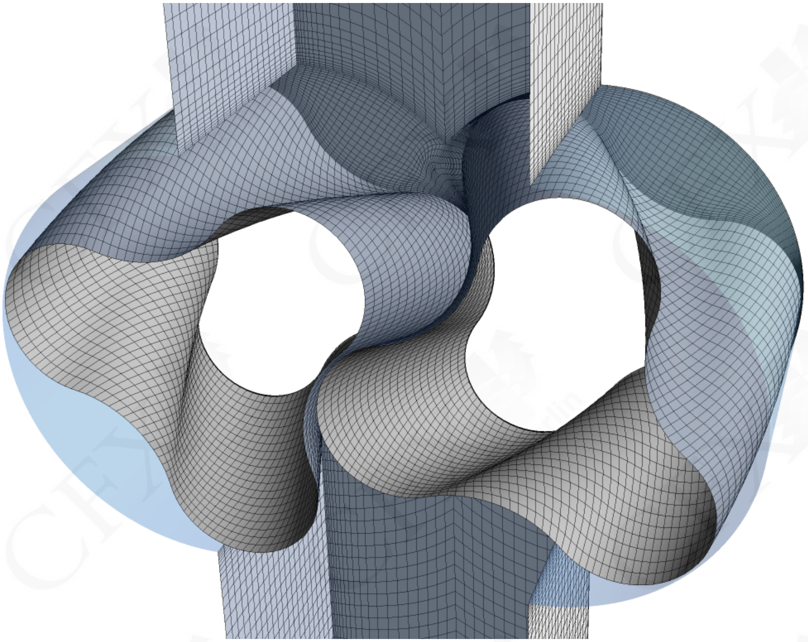

Mesh quality

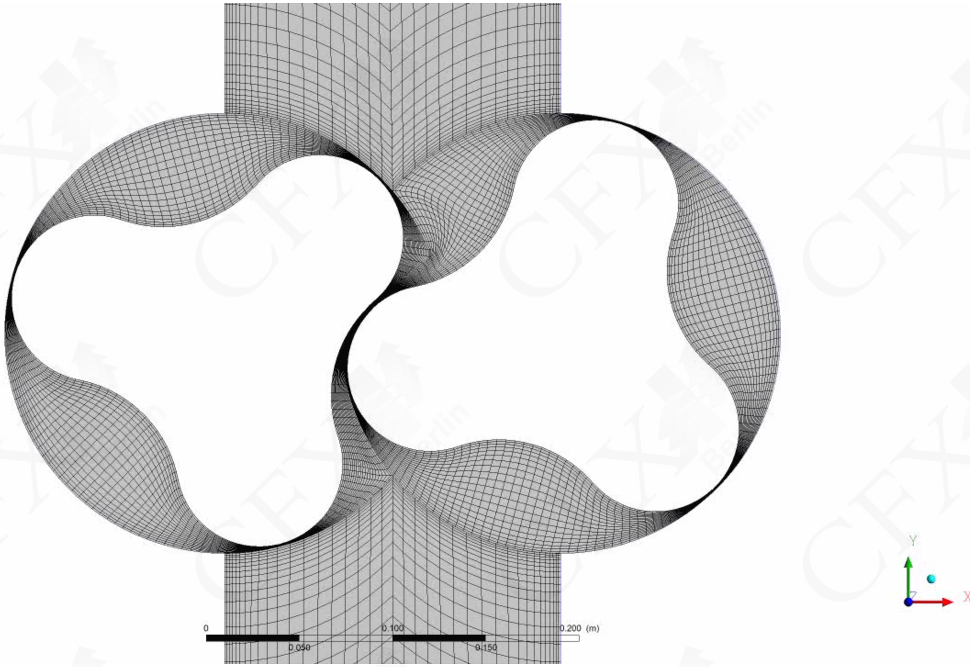
- Export
  - Mesh export to ANSYS CFX (format .cfx5) for the first rotor position
  - Export of mesh displacement coordinates for each rotor position
  - Including ANSYS CFX Session File for easy setup in ANSYS CFX



Simulation results
Lobe pumps

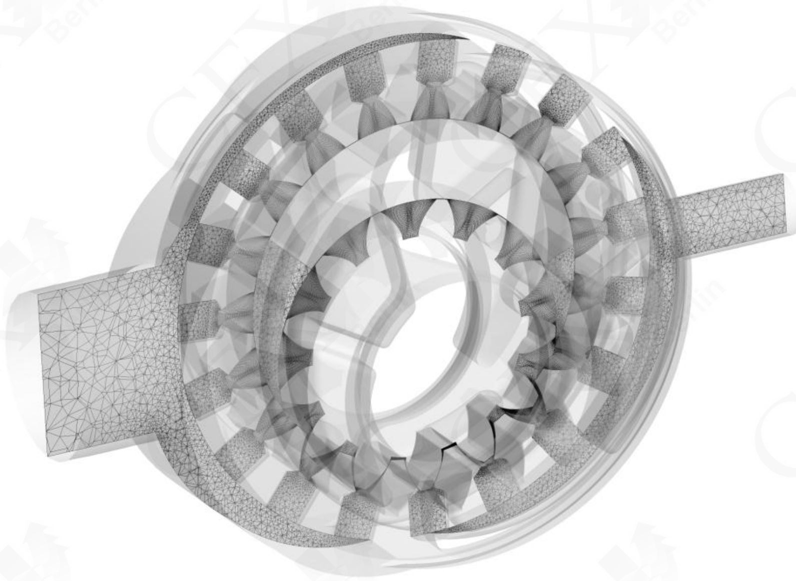
- Challenges
  - Incompressible fluids with cavitation (multiphase)
  - Non-newtonian fluids even with high viscosity

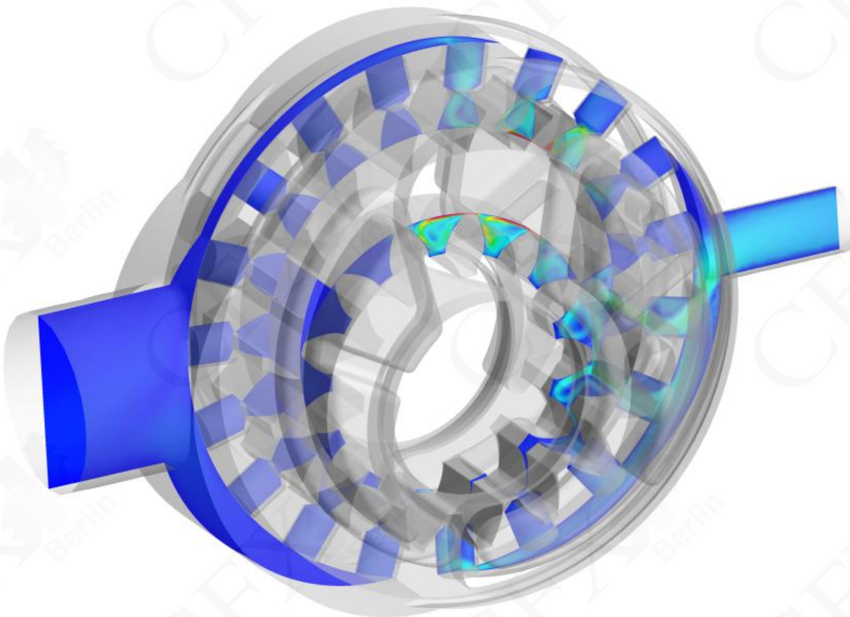




Simulation results
Gear pumps

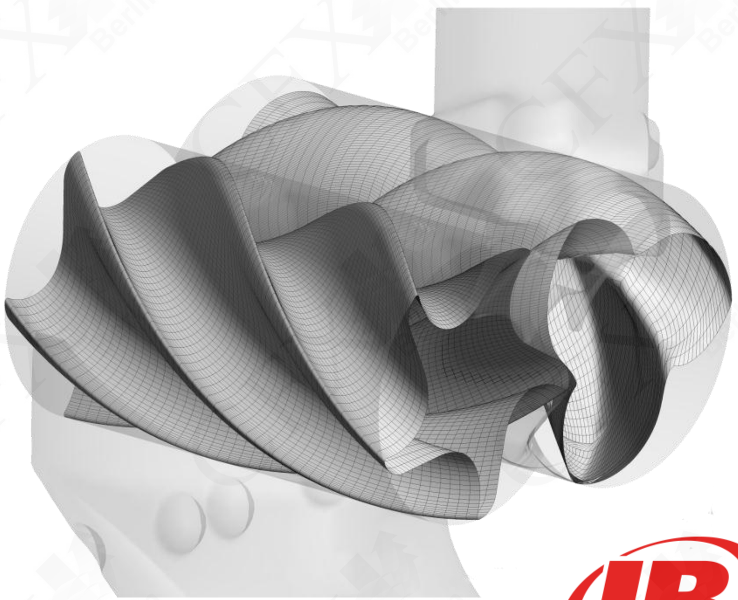
- Challenges
  - Complex rotor geometry
  - High pressure gradients
  - Incompressible fluids with cavitation
  - Non-newtonian fluids even with high viscosity

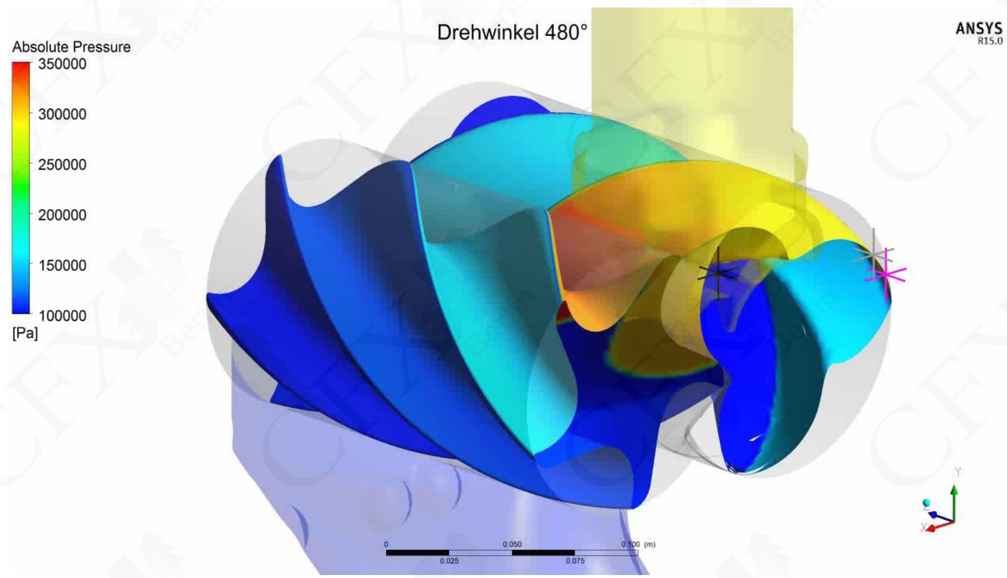




Simulation results
Screw compressor

- Challenges
  - Highly screwed rotors
  - Compressible fluids
  - High flow velocities
  - Real-gas properties could be defined
- Simulation conditions
  - Rotational speed: 13000 rpm (male)
  - Air (Ideal Gas, constant properties)
  - 2 bar pressure difference
  - Axial gap not modeled



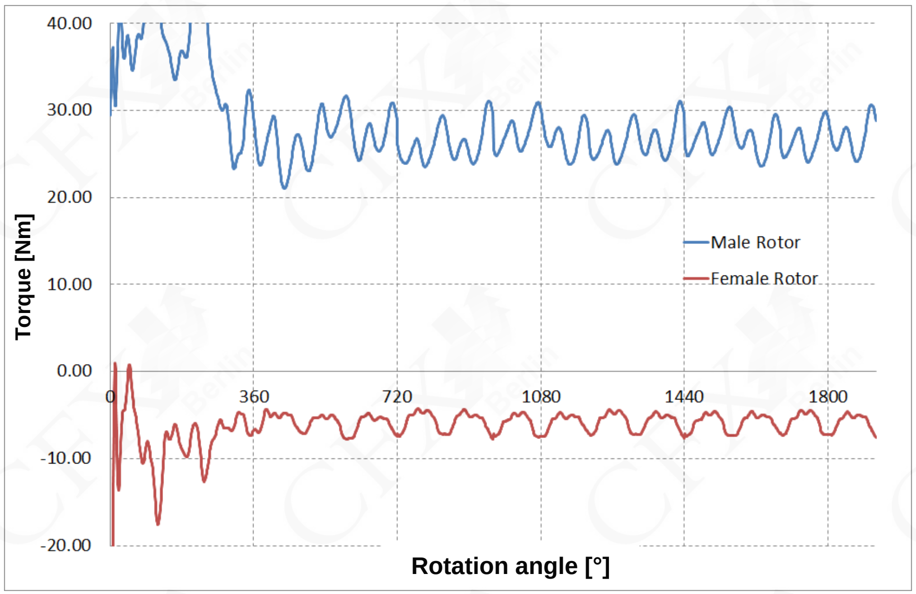


Ingersoll Rand

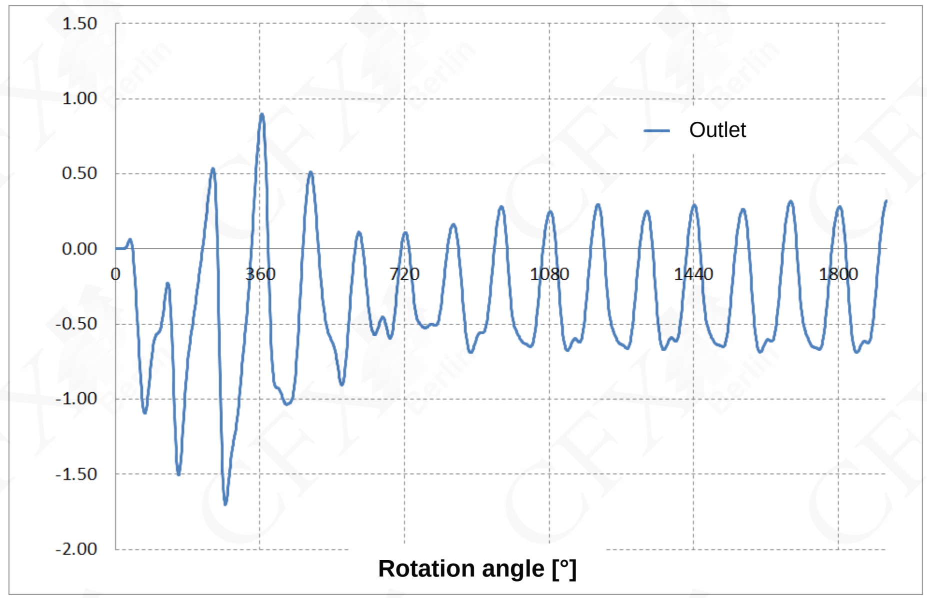
Simulation results
Screw compressor

- General analyses (Torque, Power and massflow)
  - Power: 41.2 kW
  - Averaged volume flow: 890 m³/h

Torque



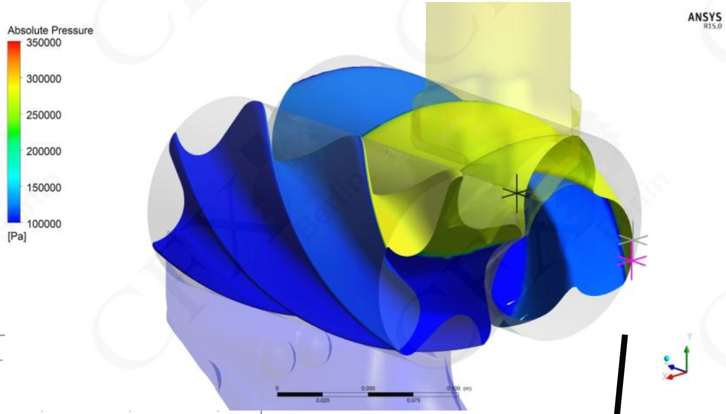
Massflow

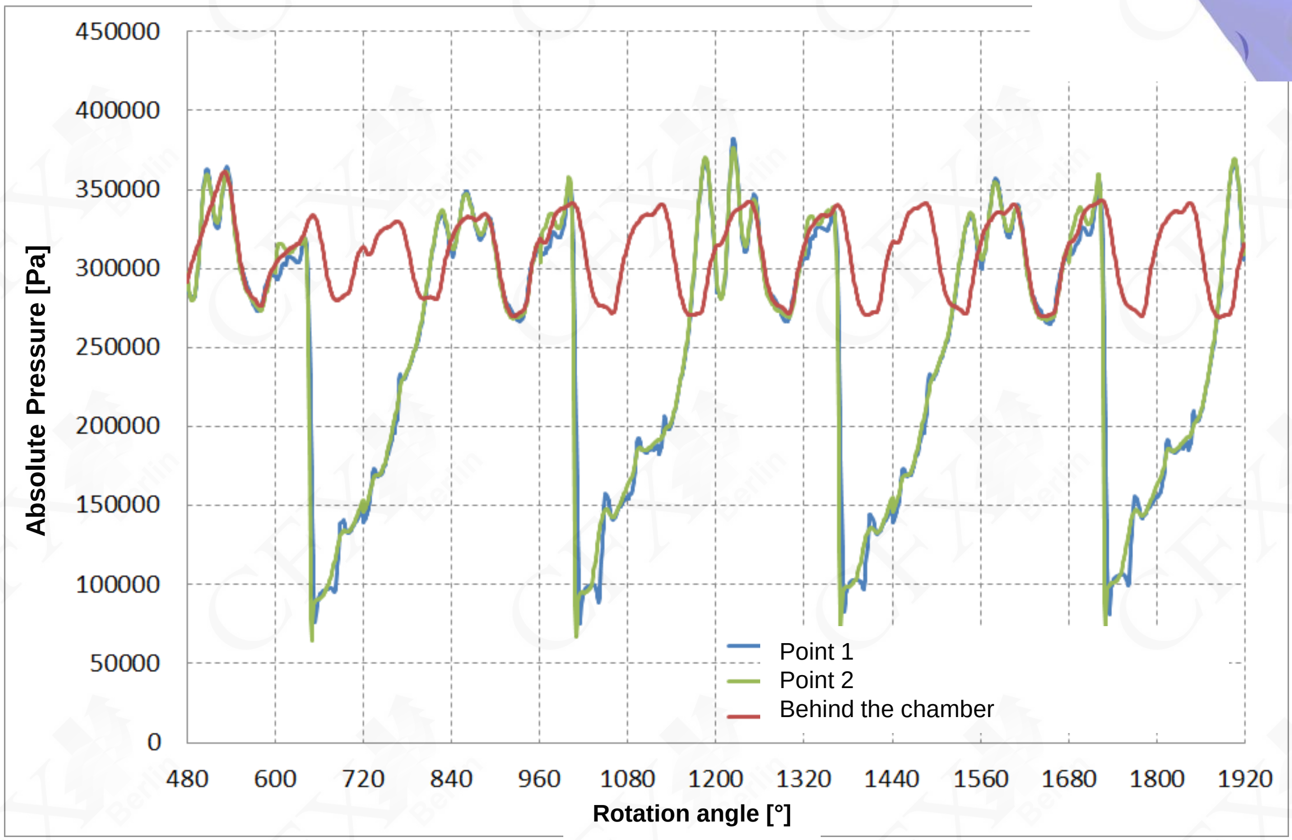


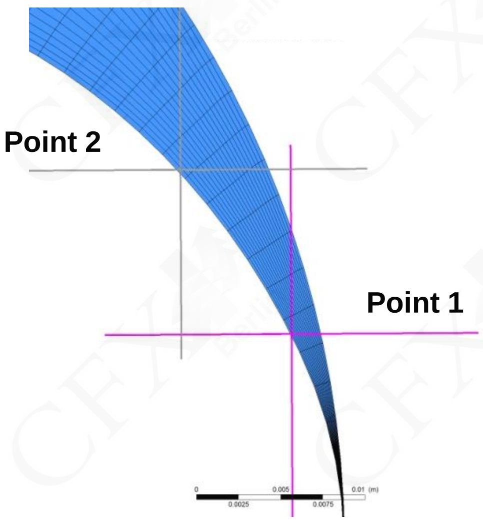
Ingersoll Rand

Simulation results
Screw compressor

- Detailed analyses (Pressure change)
  - Pressure at different measurement points

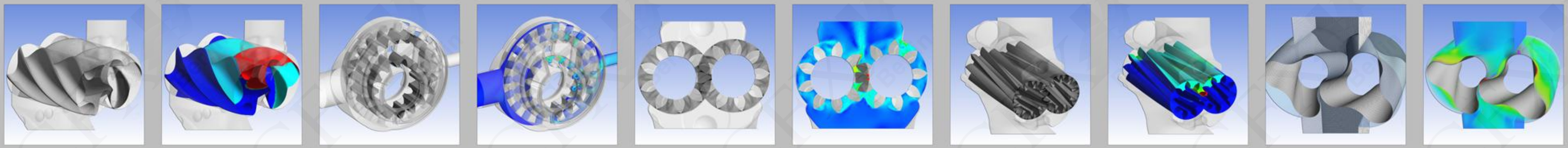




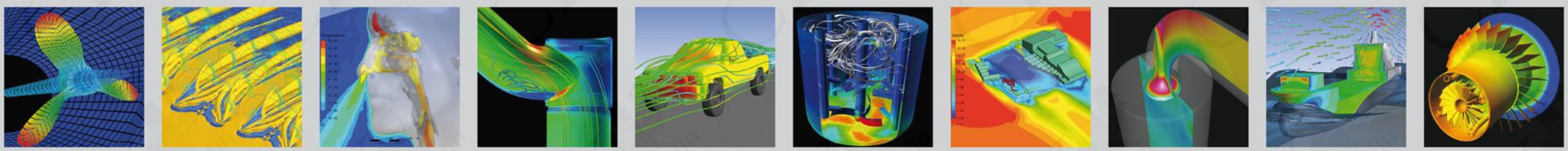


Summary

- There is now an efficient workflow for the simulation of rotary positive displacement machines with ANSYS CFD Software
- Key features of TwinMesh
  - Generation of high quality structured meshes with smoothing algorithm
  - Easy to use (comfortable GUI)
  - Works with ANSYS CFD
  - Works already for many different machine types
  - Works with gap sizes down to 1 µm
  - Individual node distribution and rotation angle steps



CFX Berlin Software GmbH



CFX Berlin

ist kompetenter Partner für:

- ANSYS Simulationssoftware
- Berechnung & Optimierung
- Beratung & Schulung
- Forschung & Entwicklung