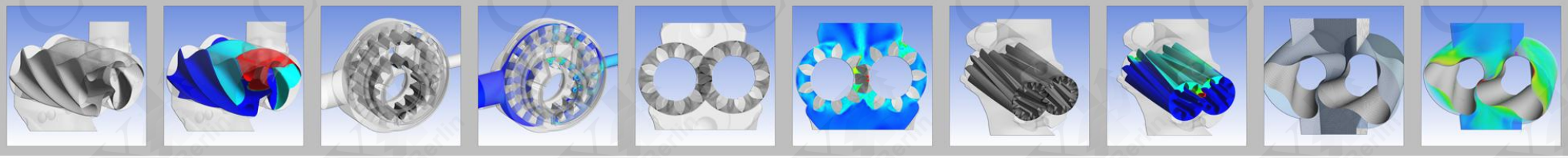


# Webinar:

## TwinMesh for Reliable CFD Analysis of Rotating Positive Displacement Machines

***14.07.2015***

**Dipl.-Ing. Jan Hesse**

*Jan.hesse@cfx-berlin.de*

© CFX Berlin Software GmbH

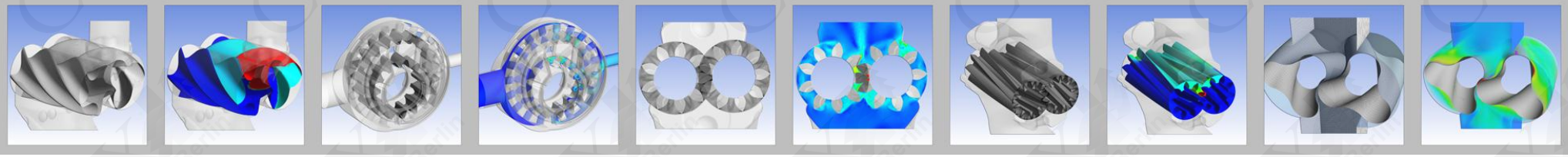
Karl-Marx-Allee 90 A

10243 Berlin

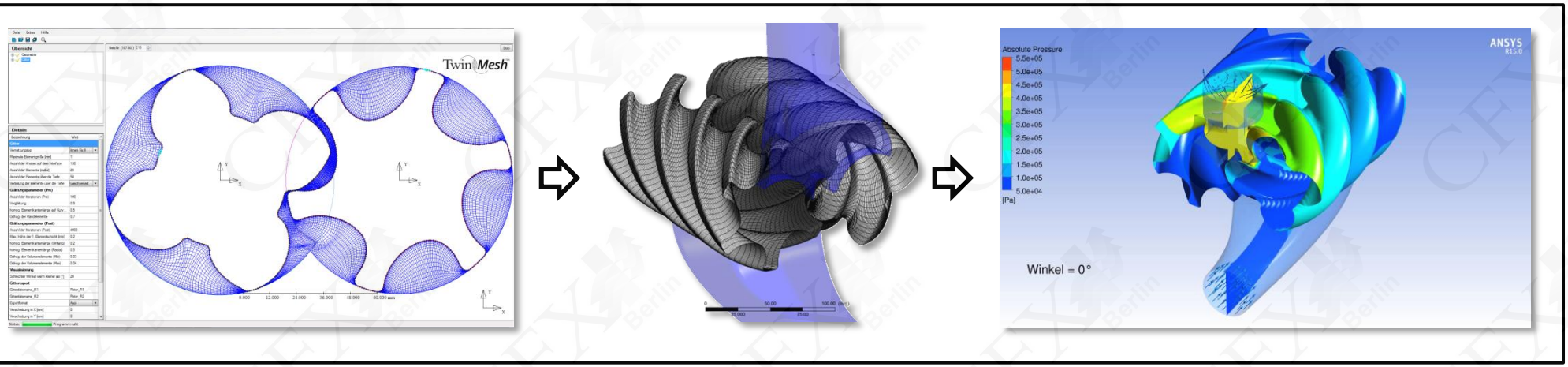
info@cfx-berlin.de | www.cfx-berlin.de | www.twinmesh.com

# TwinMesh™

CFX Berlin



- ✓ TwinMesh is a novel software that automatically generates high quality hexahedral grids for the rotary parts of positive displacement machines (developed by CFX Berlin Software GmbH)
- ✓ TwinMesh is complementary to ANSYS and allows the use of physical models available in ANSYS CFX



# CFX Berlin Software GmbH

## Excellent Simulation & Engineering Solutions

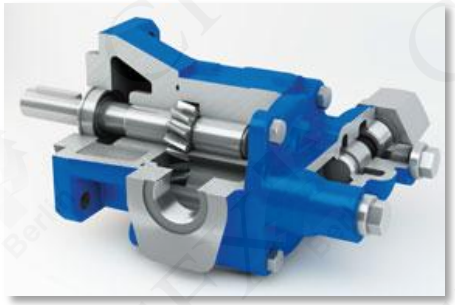
- CFX Berlin is a certified partner of ANSYS, Inc. and provides numerical simulation software and solutions for:
  - Fluid mechanics + thermodynamics
  - Electromagnetics
  - Structural mechanics
  - Coupled physics simulation

- CFX Berlin business areas:
  - ANSYS & TwinMesh software, Hardware
  - Engineering & simulation services
  - Training & consulting
  - Research & development

# Introduction

- Rotating positive displacement machines
  - Available in the current version of TwinMesh 2015

**External gear pumps**

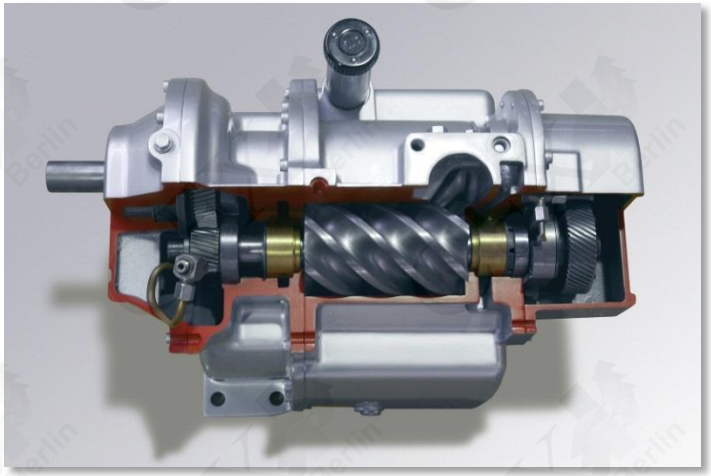


**Internal gear pumps**

**Gerotor pumps**

**Lobe pumps**

**Screw compressor**

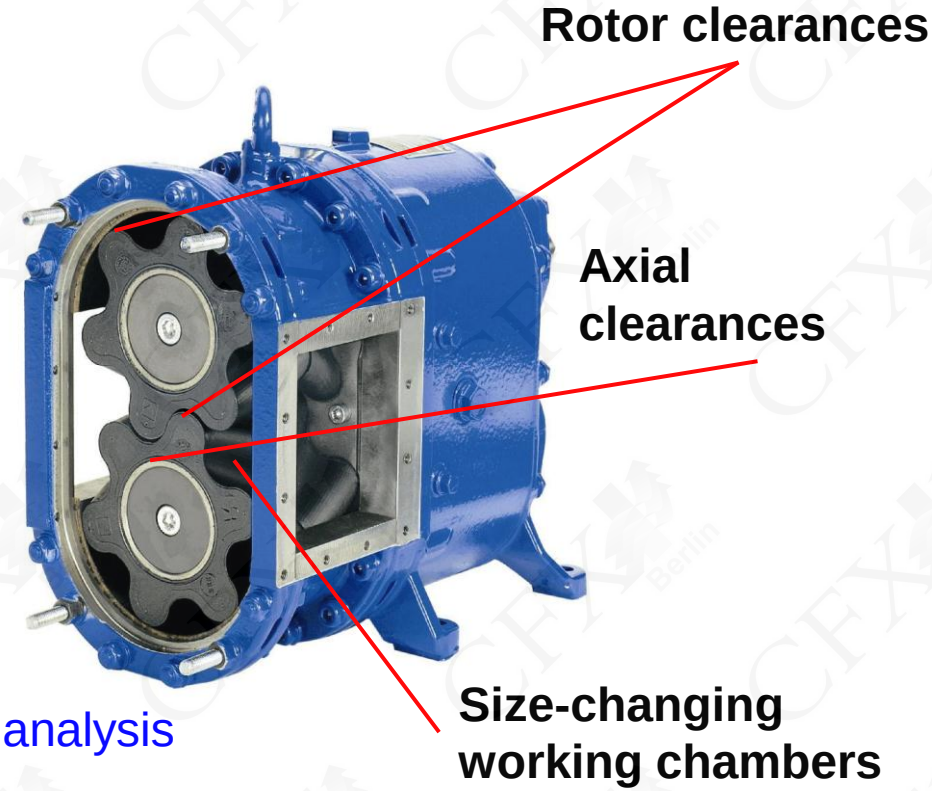


# Introduction
## What are the challenges?

- Complex geometries
  - Interlocking rotors (often screwed)
  - Size-changing working chambers with very small clearances between the lobes and between rotors and casing

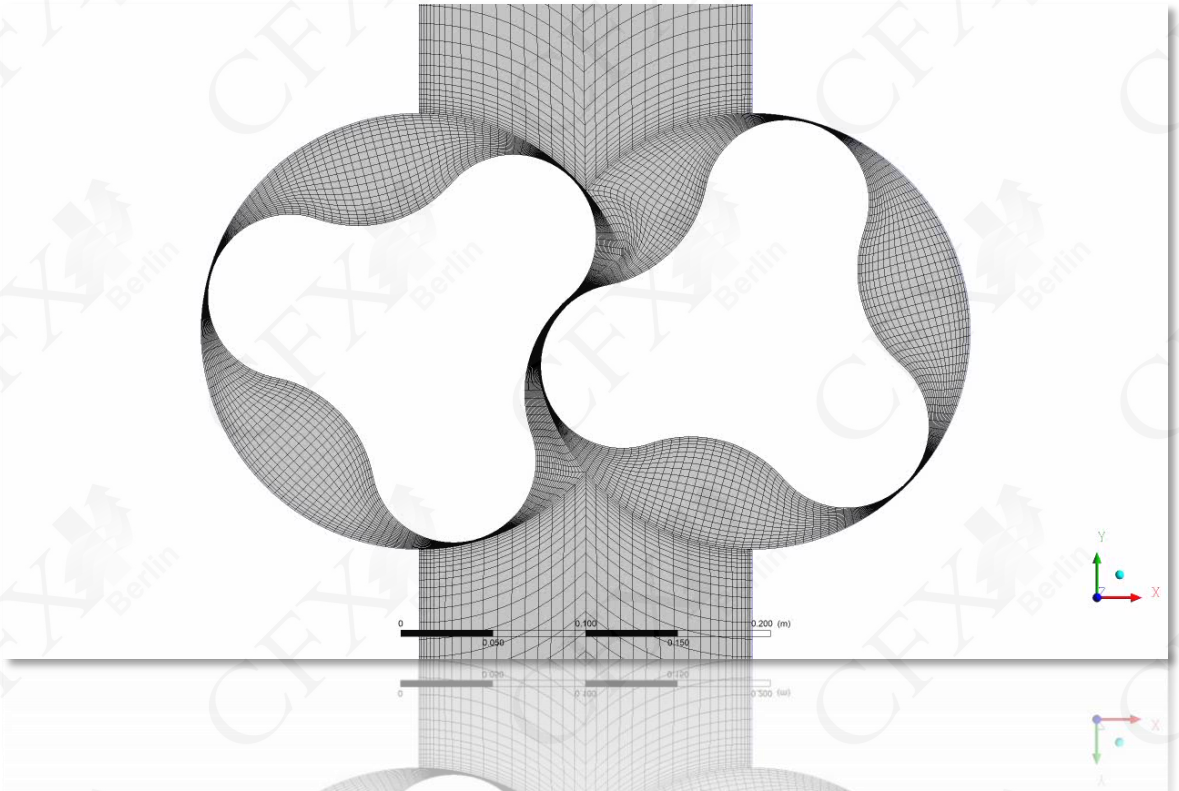
- Complex (transient) flow characteristics
  - Cavitation (Multiphase)
  - Non-newtonian fluid
  - Compressibility
  - Real-gas properties
  - Turbulence
  - Viscous heating, etc.

- These challenges lead to **specific meshing requirements** for reliable CFD analysis
  - Min. angle, volume change, aspect ratio



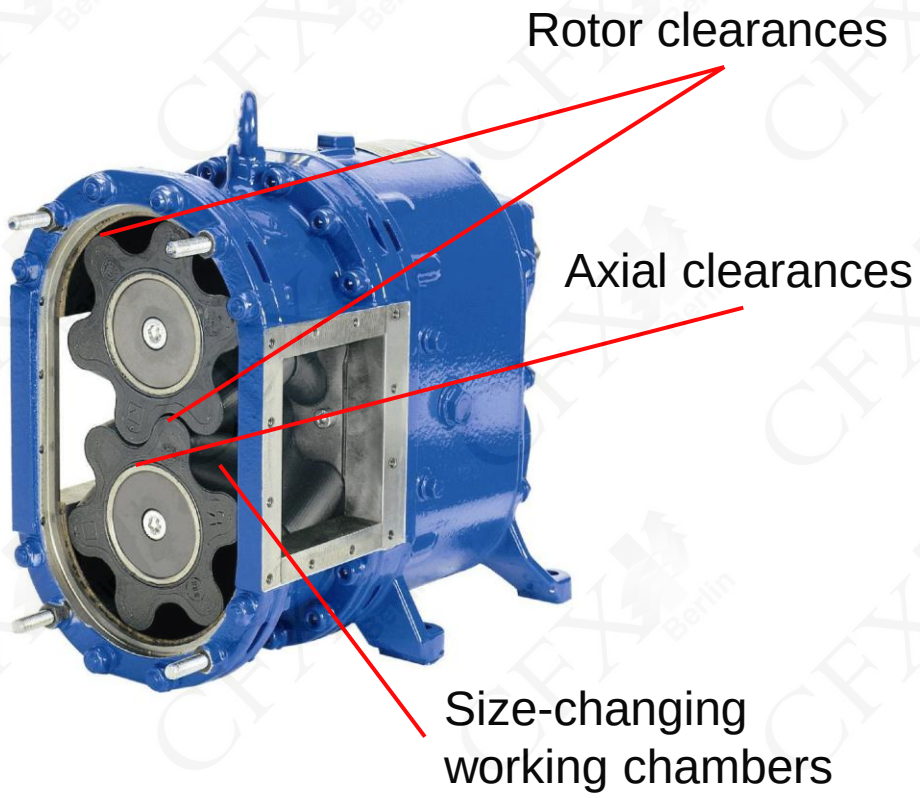
# Introduction
## Realization in a CFD simulation

- Time-depending change of the flow volume inside a lobe pump
  - Realized with TwinMesh 2015 and ANSYS CFX



## Overview

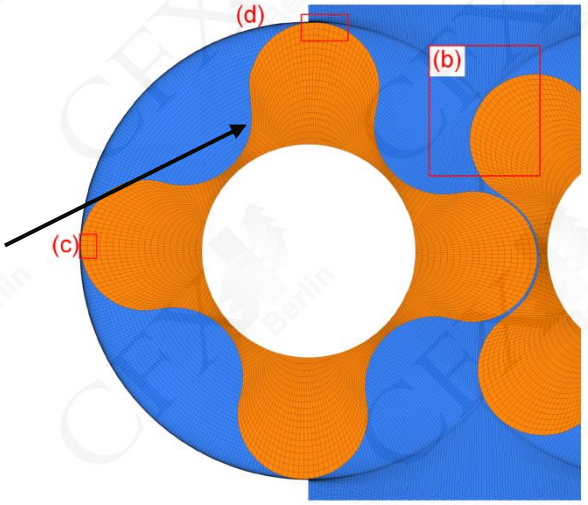
Methodes for chamber modelling without TwinMesh



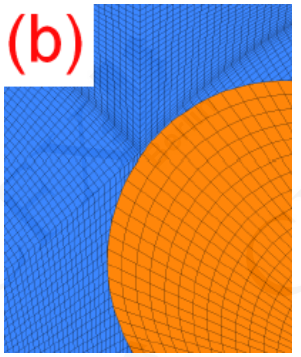
# Methods for chamber modelling
## Immersed-Solid

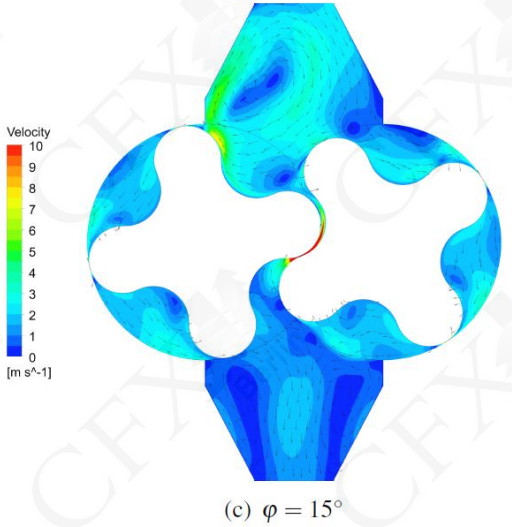
- Advantages
  - Fast – only one mesh for fluid and solid volumes
  - Variable time step control

- 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



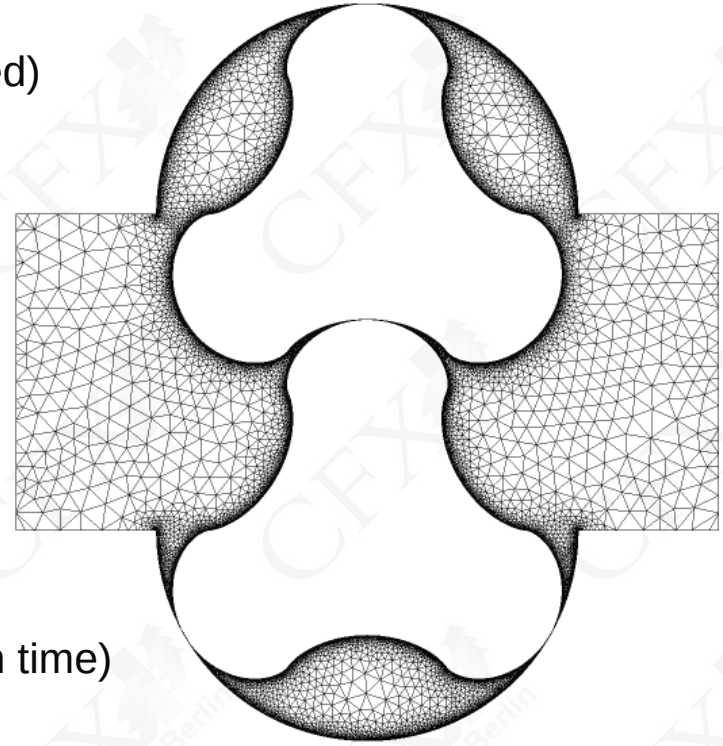




(c) $\varphi = 15°$

**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)

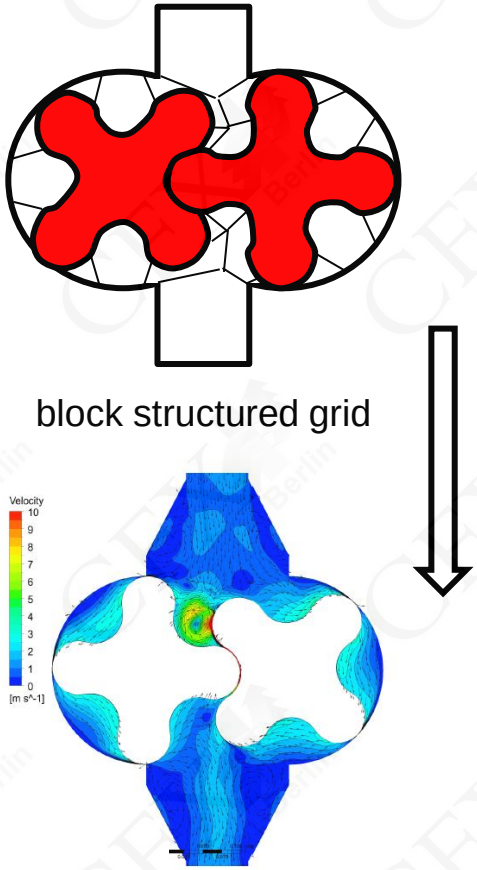


- Disadvantages
  - Mesh-generation for almost each iteration
    (increasing computation time)
  - Element topology (Tetra) leads to **very high element numbers in gaps** (increasing computation time)
  - **Mesh quality issues** due to mesh deformation and element topology when using remeshing
  - **Numerical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

# Methods for chamber modelling

## Manual Generation of Structured Hexahedral Meshes

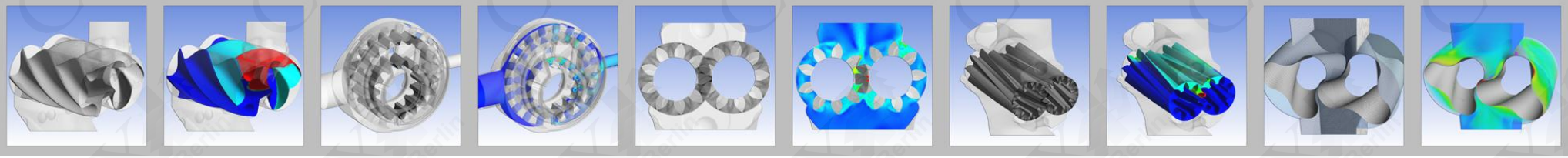
- Idea
  - Manual grid generation in ANSYS ICEM CFD Hexa for many rotor positions per rotation
- Advantages
  - Best mesh and numerical quality
  - High resolution of gaps is possible
  - Element topology allows manageable model size
  - No interpolation errors since the grid topology remains the same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



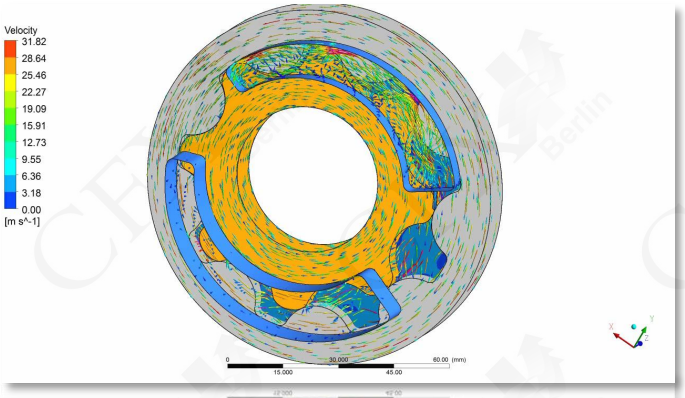

**Fuchs, M.:** „Numerische Simulation der instationären Strömung in einer Drehkolbenpumpe“, Bachelorarbeit, Technische Universität Berlin, 2010

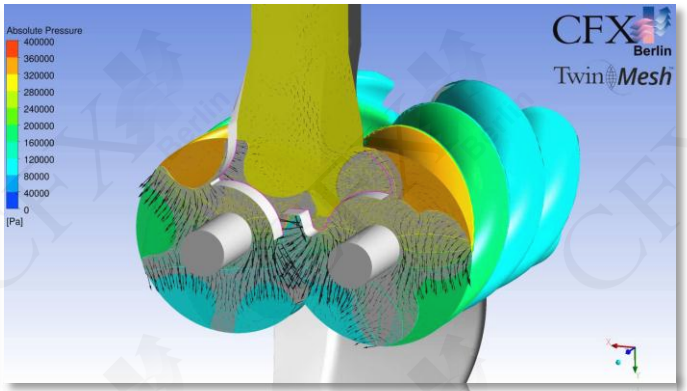
# Methods for chamber modelling
## TwinMesh – an essential part of CFD

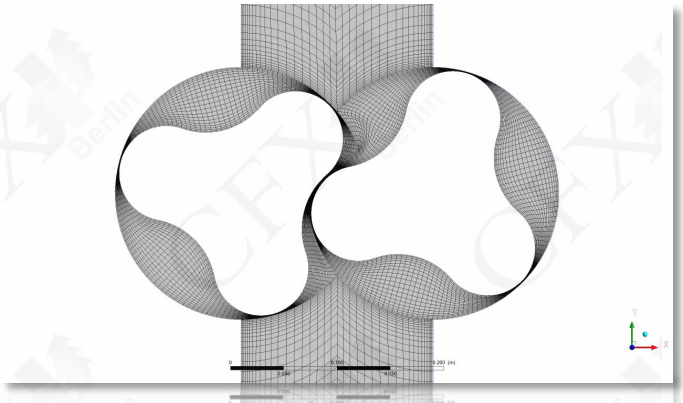
- TwinMesh was developed based on the manual approach by using structured hexahedral meshes including the following specifics
  - **Fast mesh generation** with less manual effort
    - reduce engineering working time
  - Generation of **high quality structural meshes** for **high quality CFD results**
    - Check quality before simulation
    - Reliable simulation results in comparison with measurement data
  - Provide **templates** to speedup the CFD workflow
    - ANSYS CFX Setup files for each type of machine (only change the mesh and the boundary condition values to finish the setup)
    - Session file to transfer information from TwinMesh to the CFD setup
  - TwinMesh meshes **work with ANSYS CFD**
    - Use of  high quality numerical models



# How does TwinMesh work?

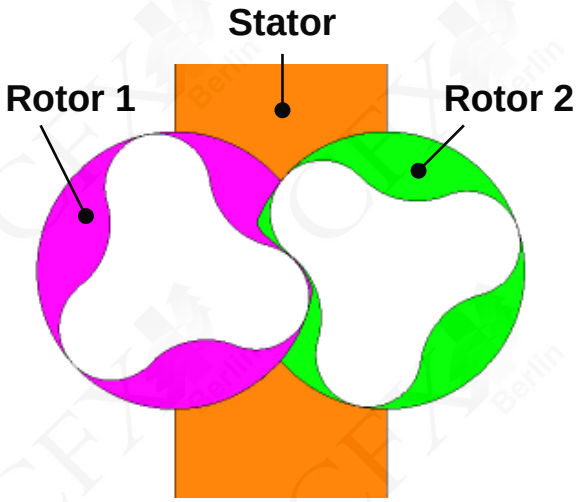


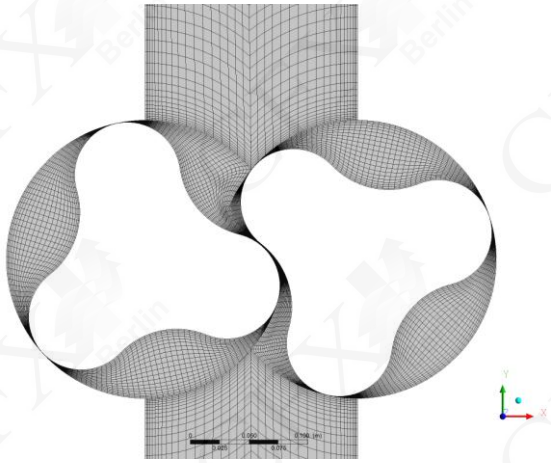



- TwinMesh is a novel software, developed by CFX Berlin Software GmbH which generates high-quality hexahedral meshes for the rotating parts of axis parallel rotary positive displacement machines.

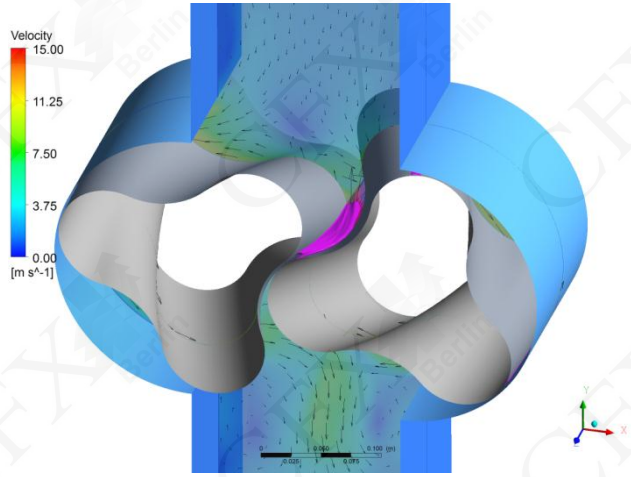
## Simulation Workflow




Split of the simulation domains into steady and rotating parts

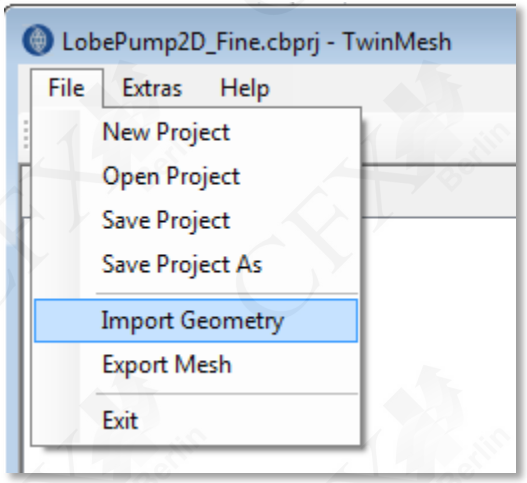


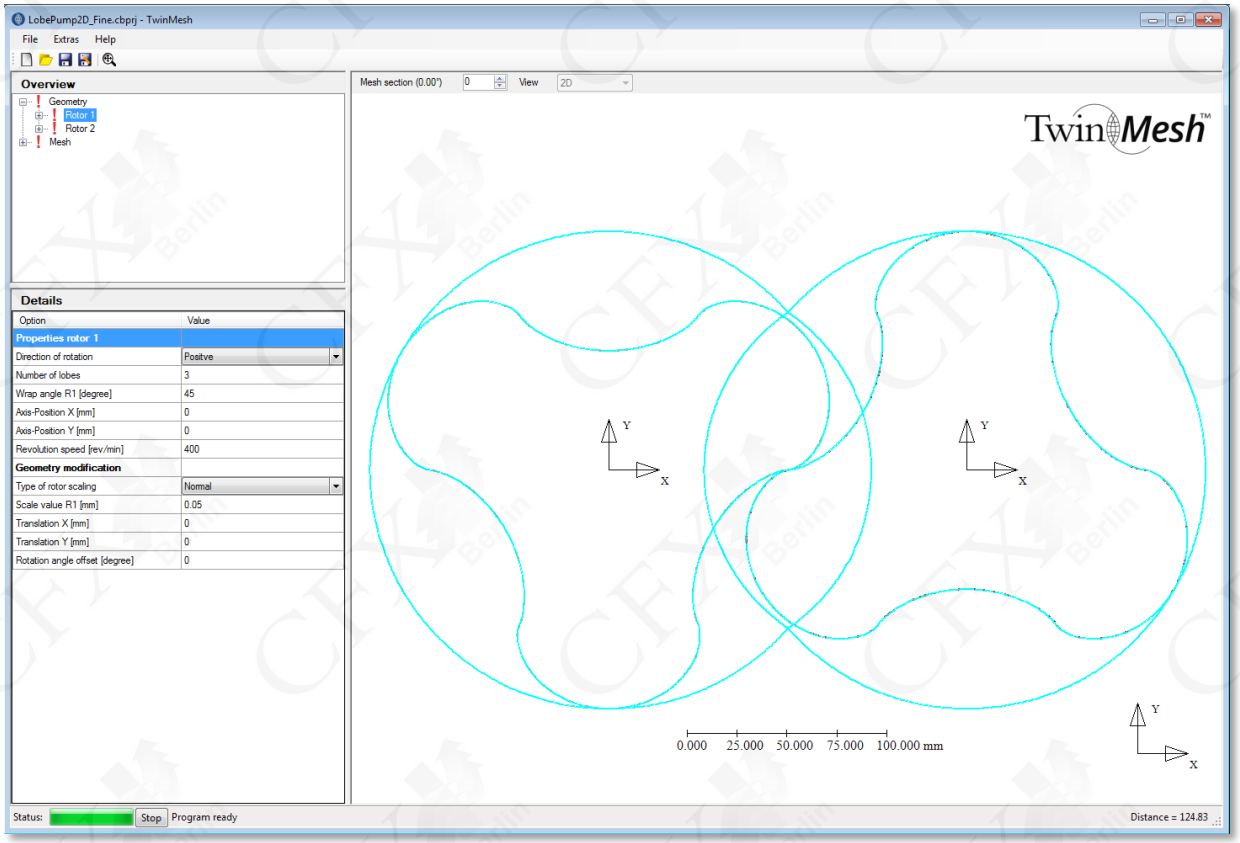
Grid generation for the rotors with **TwinMesh** and for the Steady parts with ANSYS ICEM CFD or ANSYS Meshing



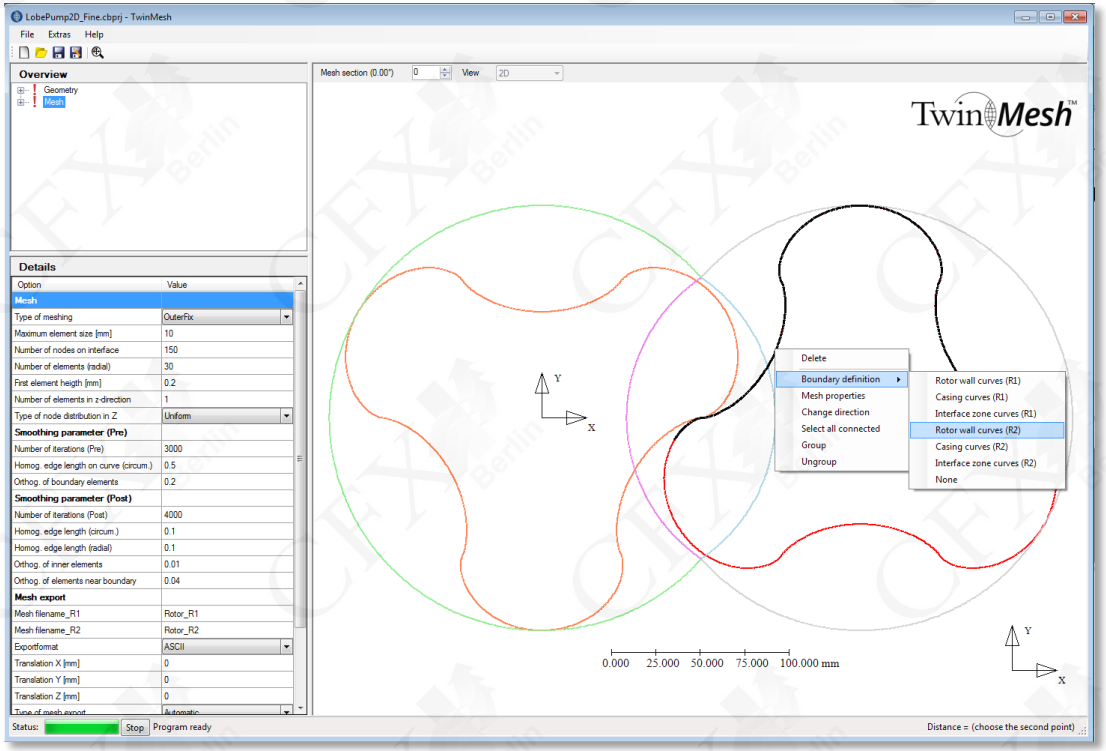

Numerical calculation with ANSYS CFD

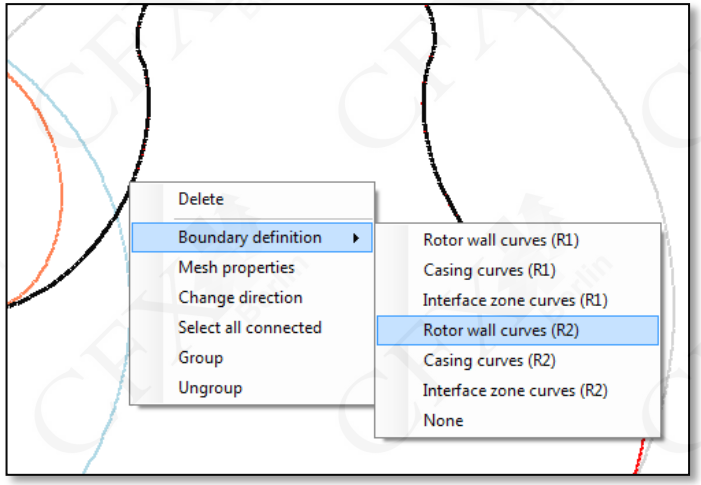
- File format (2D cross section of the machine)
  - IGES
  - CSV-File with point coordinates



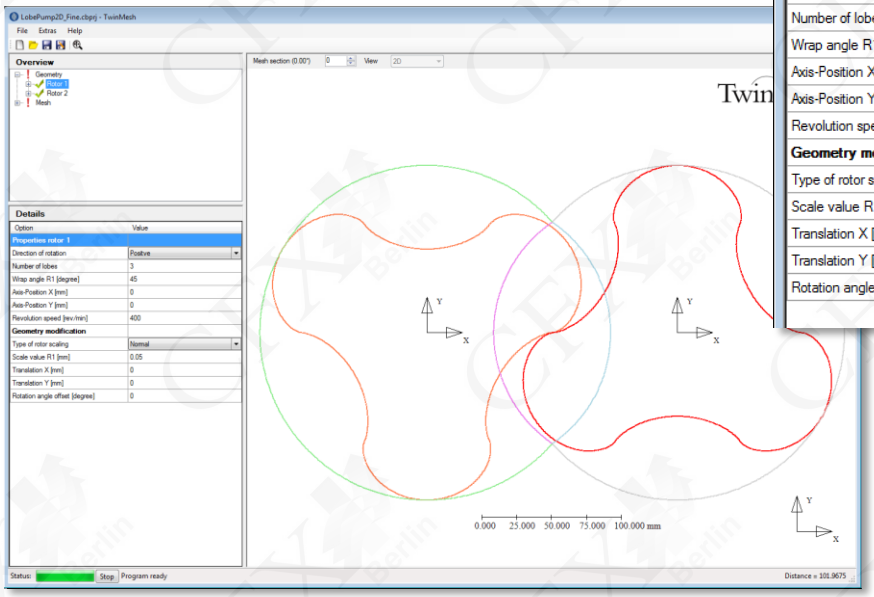



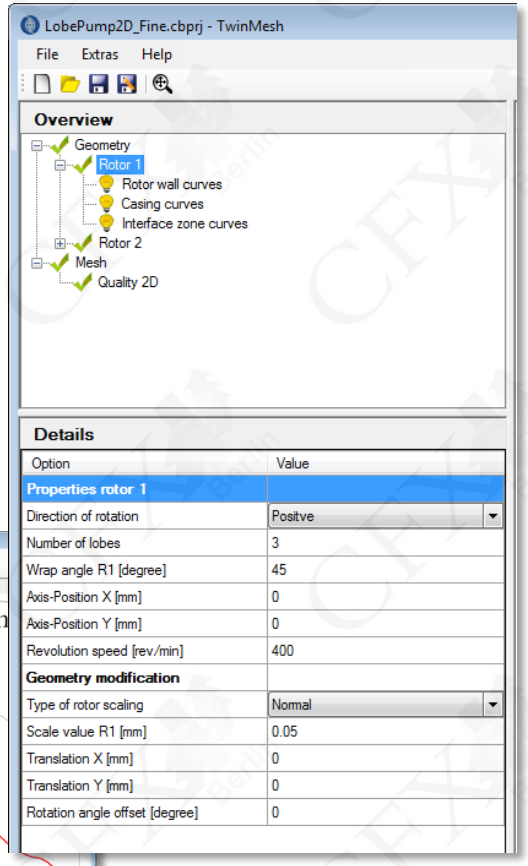
- Boundary types
  - Rotor curvature
  - Casing curvature
  - Additional curvature for interface creation

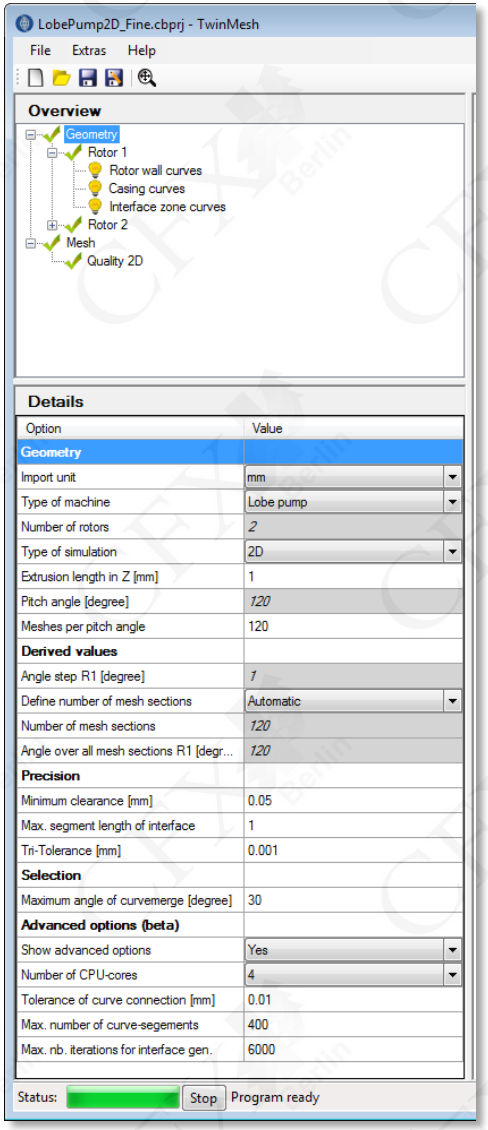


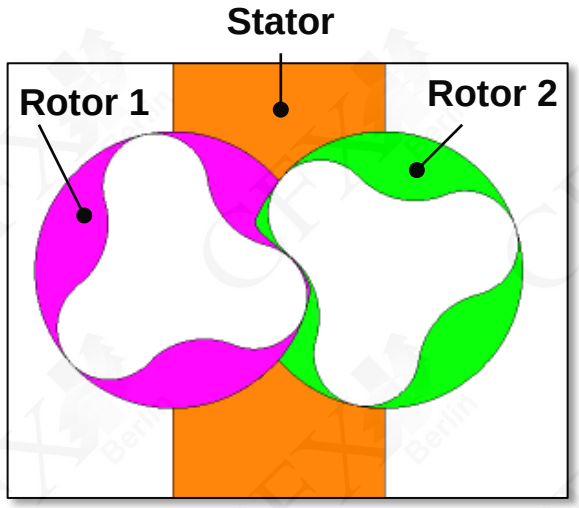
- Options
  - Type of machine
  - Number of lobes
  - Rotorposition
  - Rotor curvature modifications
  - Screwing
  - Meshes per pitch angle
  - …

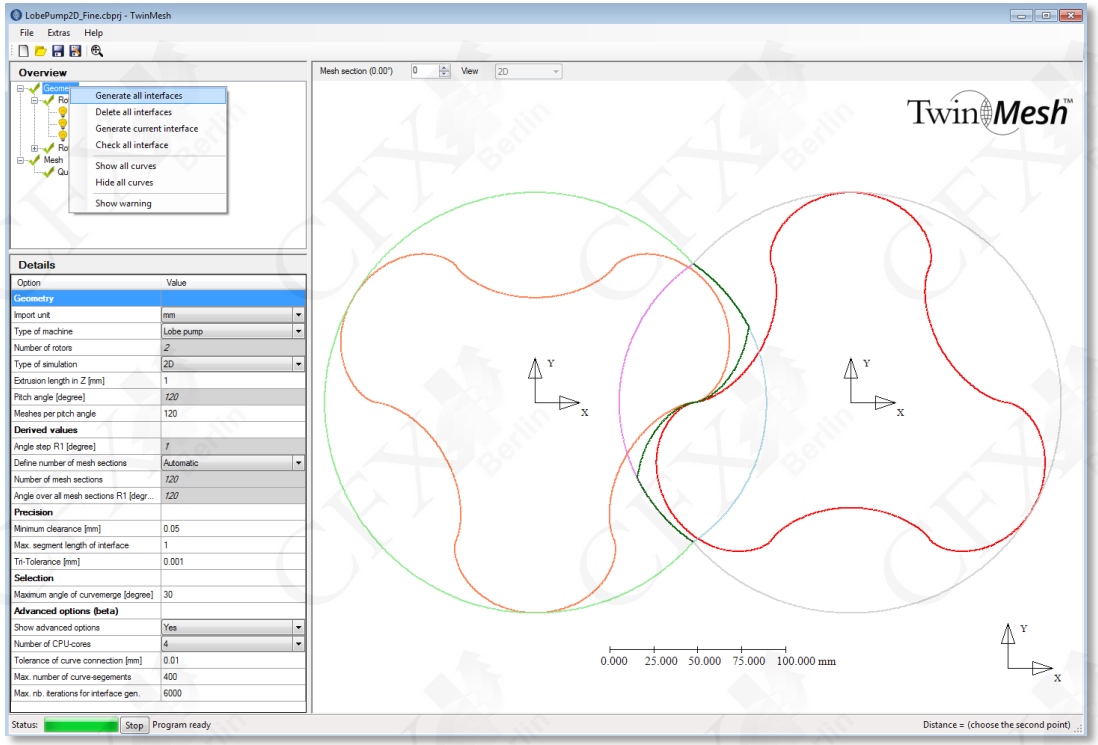


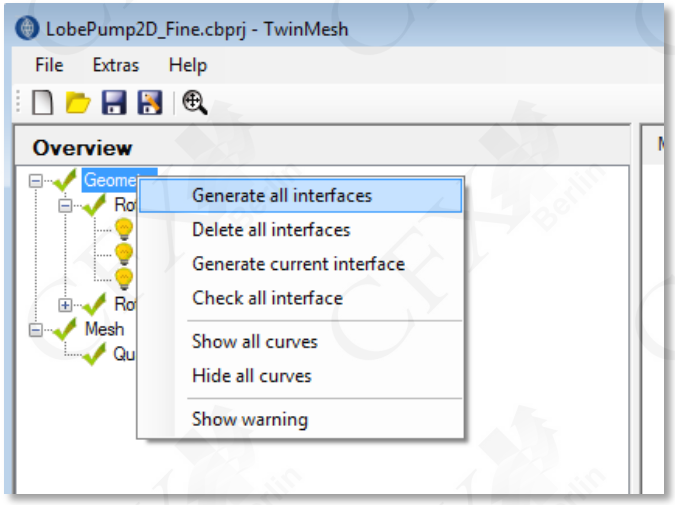


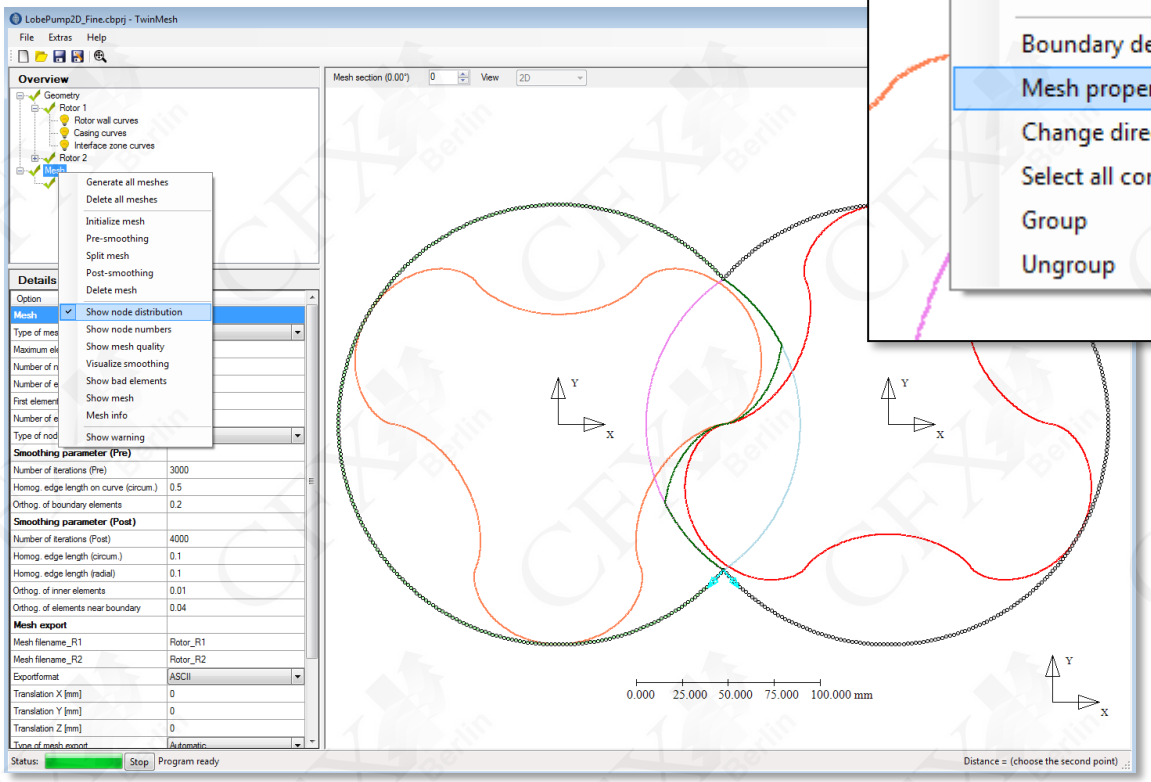
- Automatic contact definition for the rotor meshes
  - Interfaces between the rotor meshes are automatically generated depending on rotor curvature

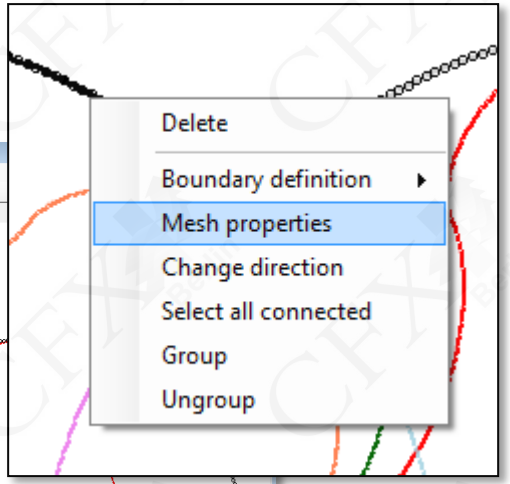


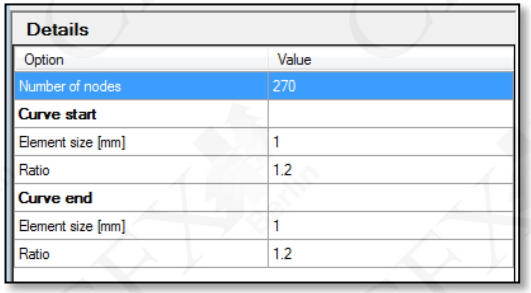



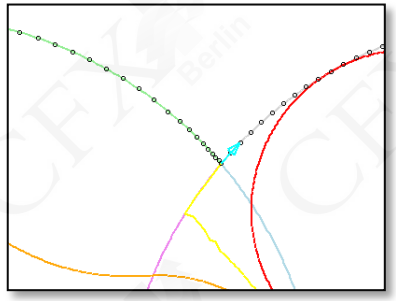



- Definition of node distribution
  - On curves
  - In radial and axial direction
  - Inflation layer

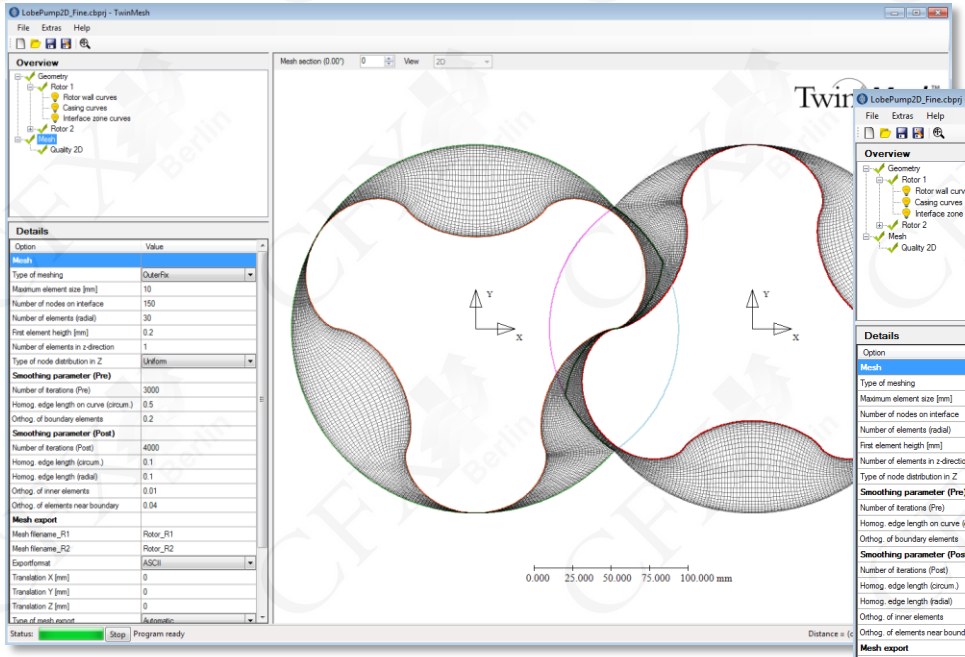


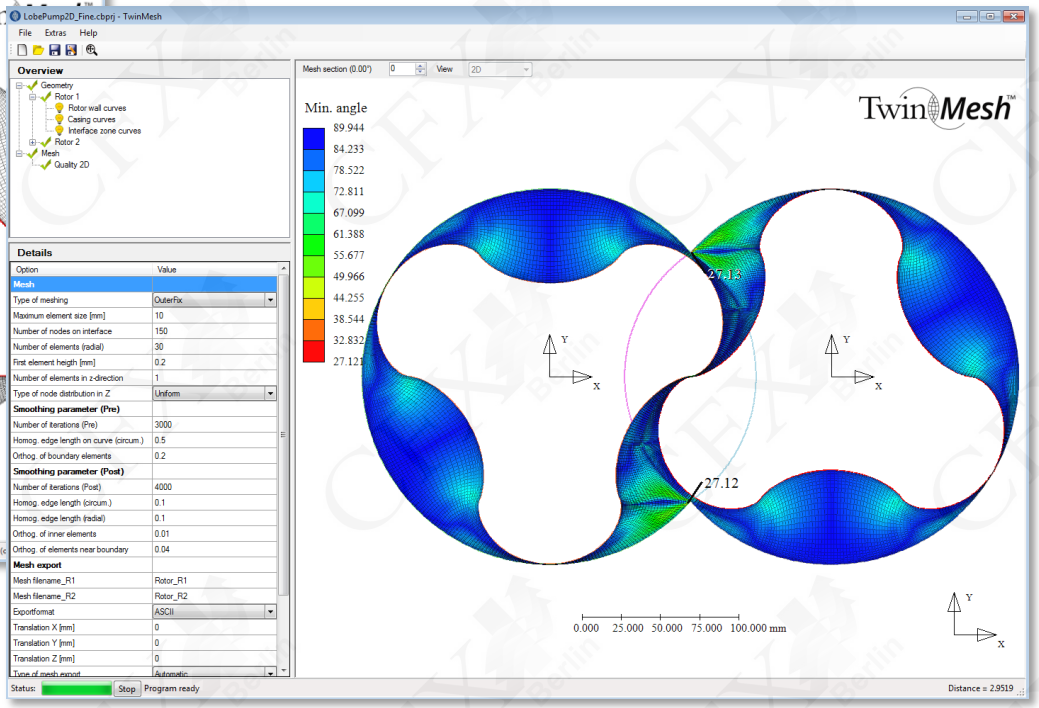



- Automatic mesh generation for each rotation angle
  - Smoothing algorithm depending on orthogonality and volume change
  - Different methods of mesh connection at the rotor-rotor-interface available (non-conforming or 1to1)
  - Visual and quantitative quality check tools available

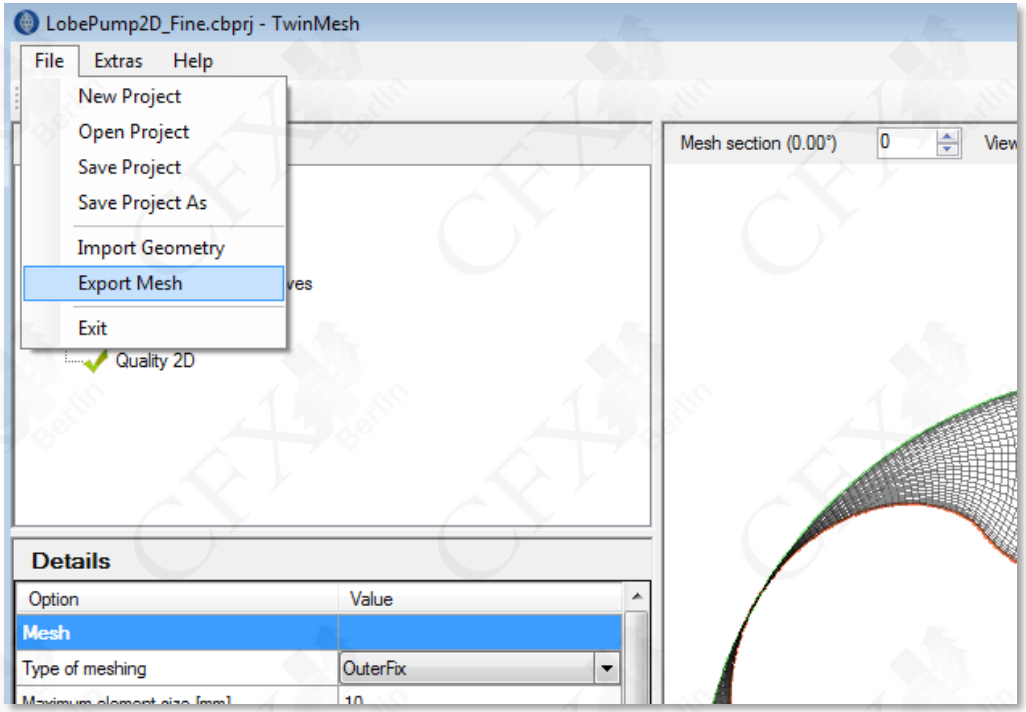



**Mesh**

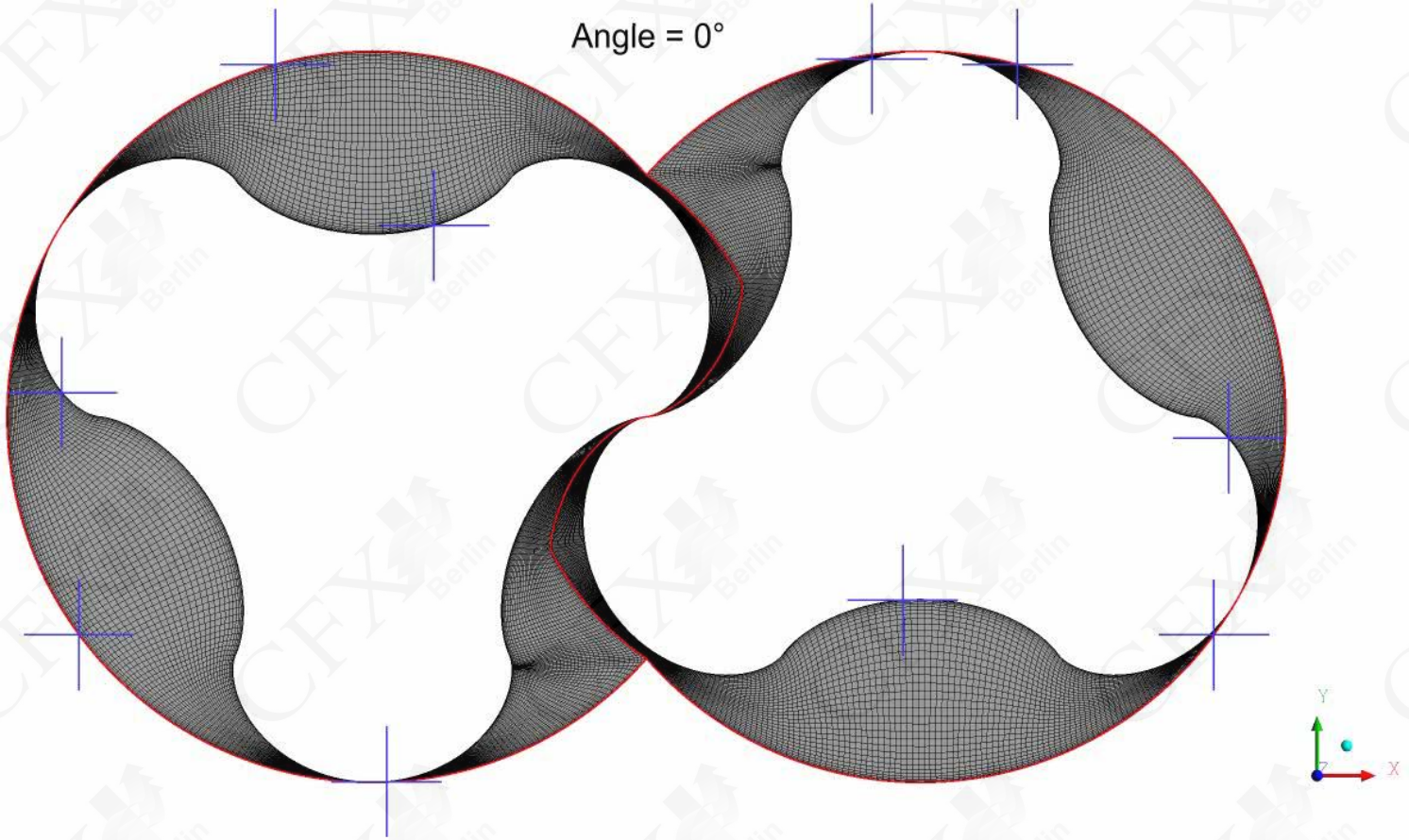
**Mesh quality**



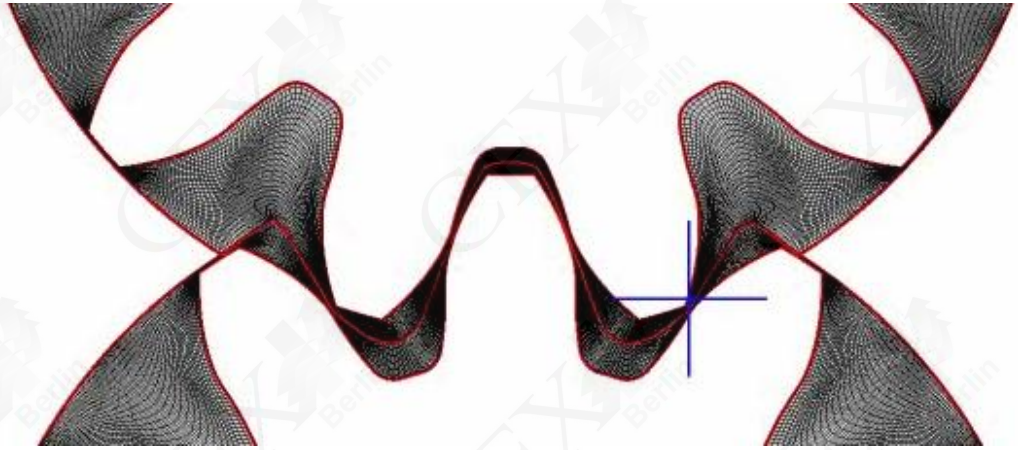
- Export
  - Mesh export to ANSYS CFX (format .cfx5) for the first rotor position
  - Export of mesh displacement coordinates for each rotor position
  - Including ANSYS CFX Session File for easy setup in ANSYS CFX



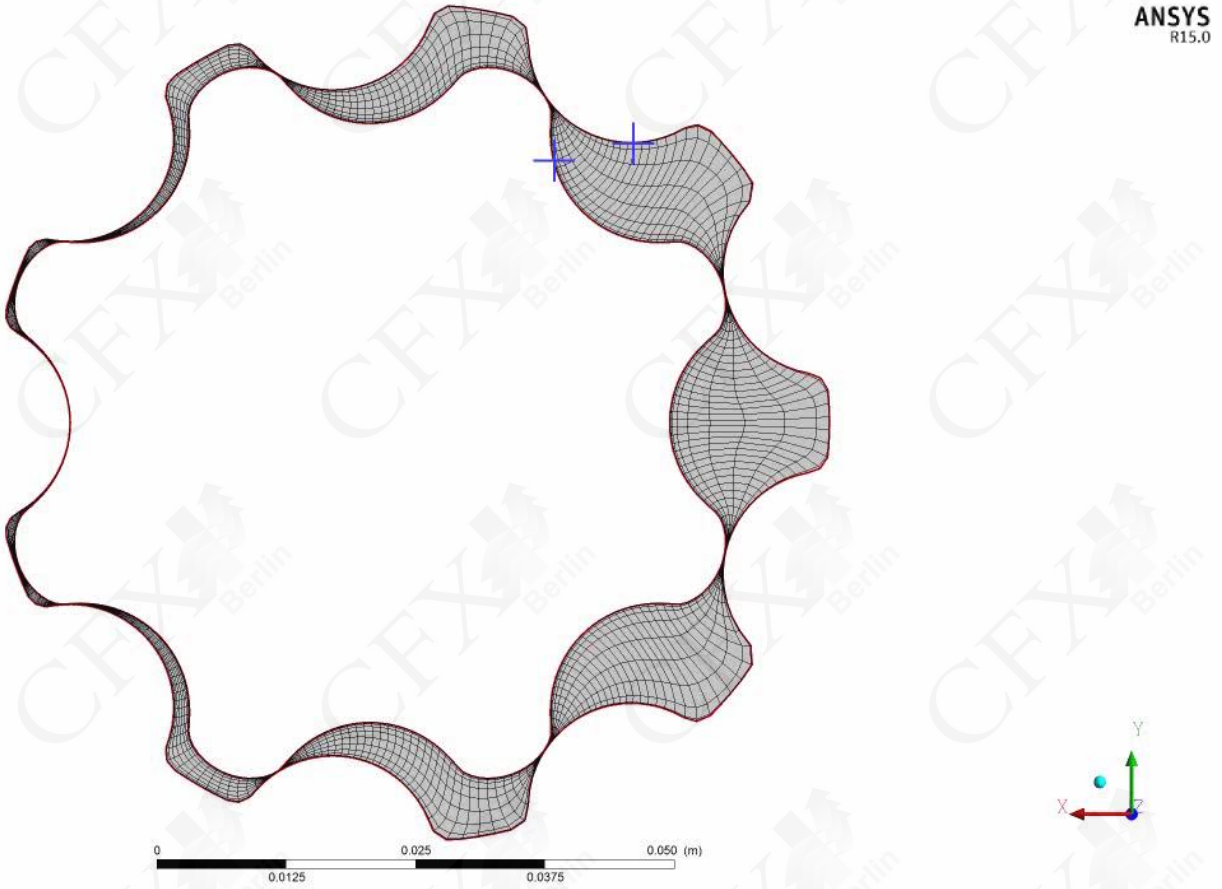
- Mesh movement of a lobe pump
  - with the “OuterFix”-type strategy



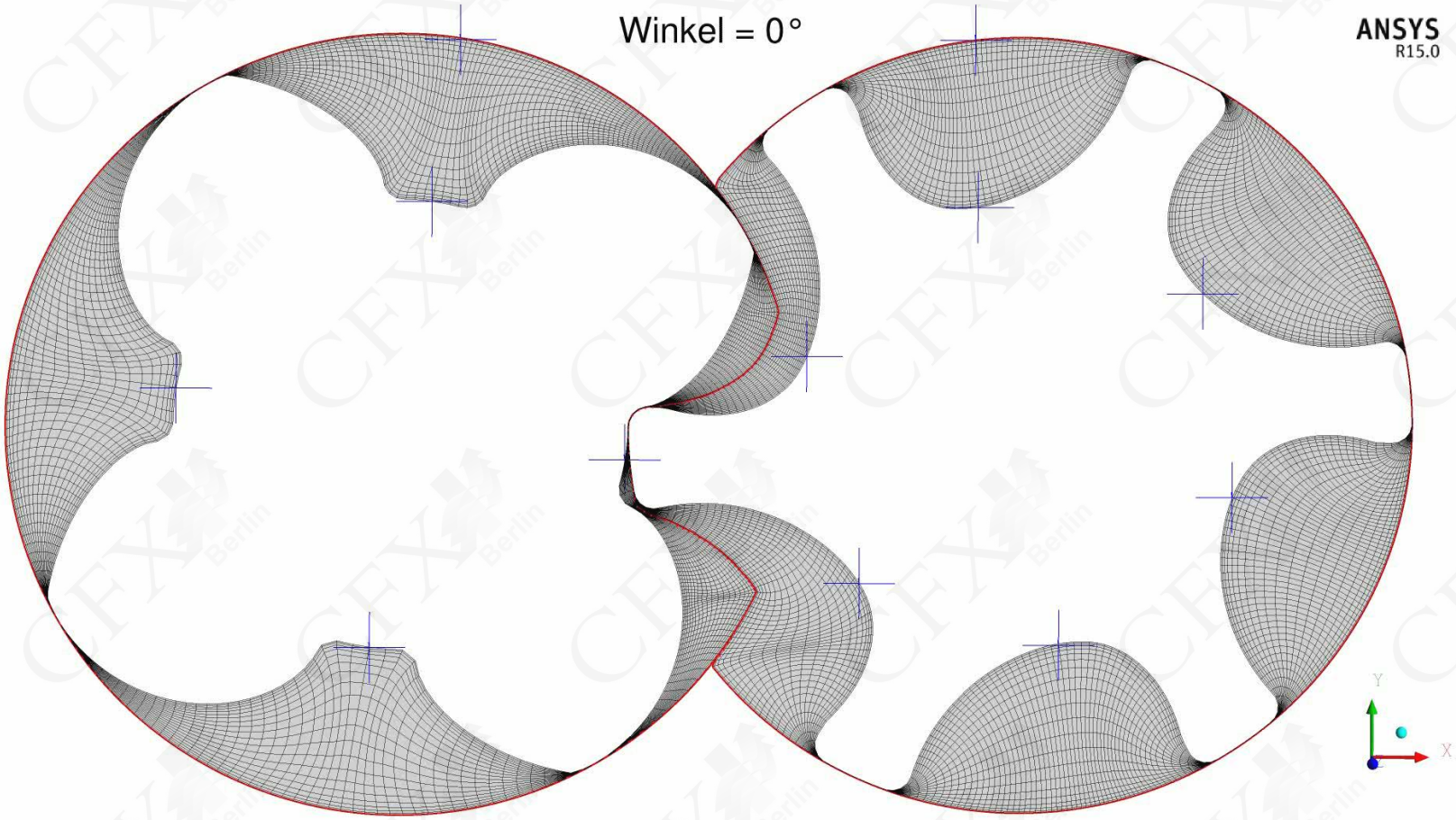
- Mesh movement of an external gear pump
  - with the “InnerFix”-type strategy



- Mesh movement of a gerotor pump
  - with the “InnerFix”-type strategy

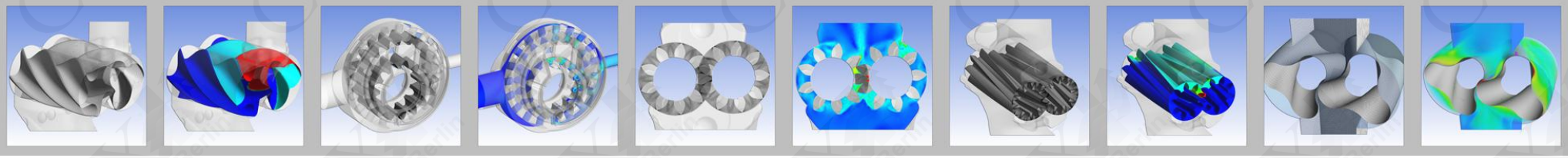


- Mesh movement of a screw compressor
  - with the “Mixed”-type strategy

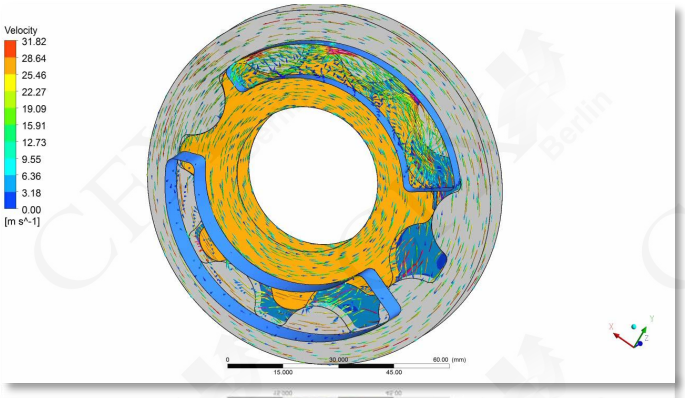


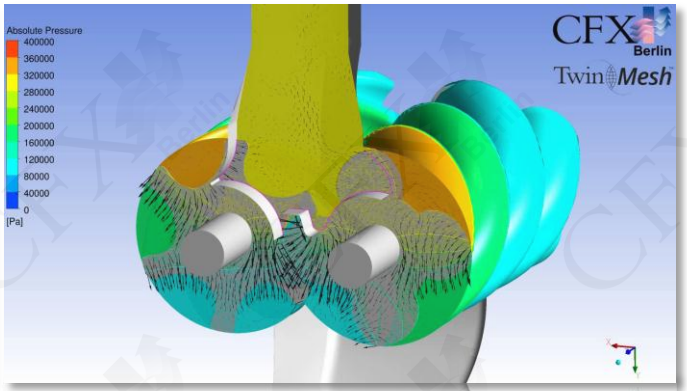
TwinMesh™

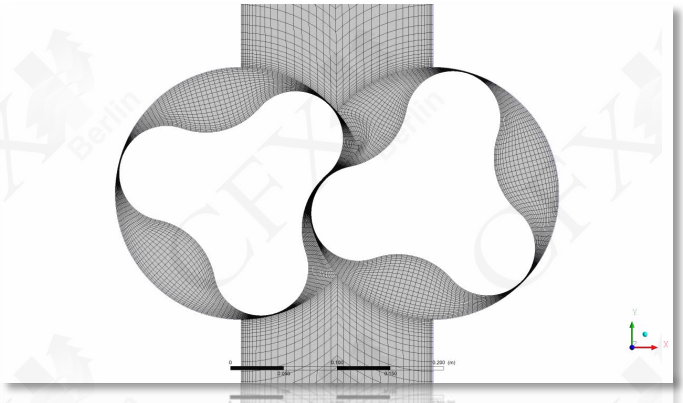
CFX Berlin



# CFD simulation examples for rotating pd machines



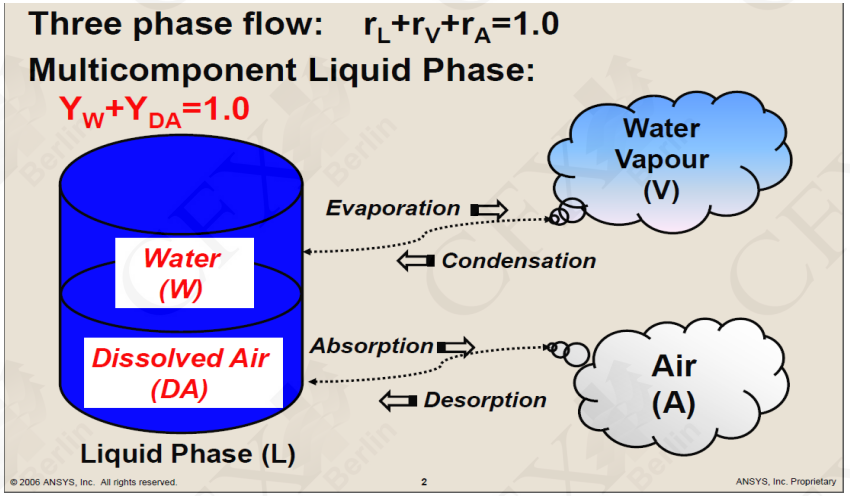



# Simulation examples
## External Gear Pump

- Flow characteristics
  - High pressure gradients
  - Full cavitation model
    - Vapour cavitation and aeration
  - SST- Turbulence model

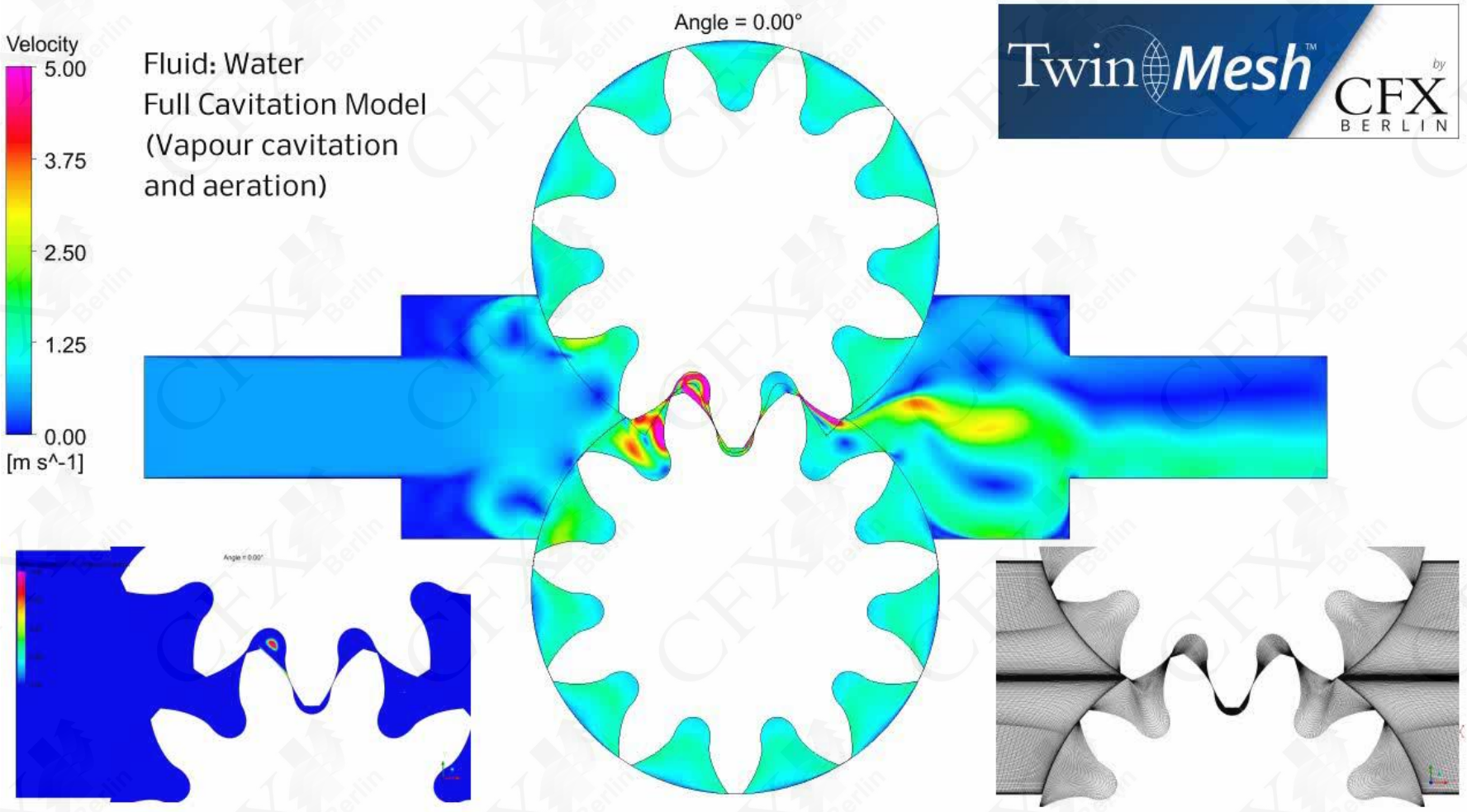
- Boundary conditions
  - Inlet
    - 1 bar (abs)
  - Outlet
    - 11 bar (abs)
  - Rotational speed
    - 500 rev/min

- Output
  - Time dependent information
    - Pressure pulsation
    - Flow velocities
    - Cavitation behavior
  - Average information (also transient available)
    - Mass flow
    - Torque on rotors
    - Temperature



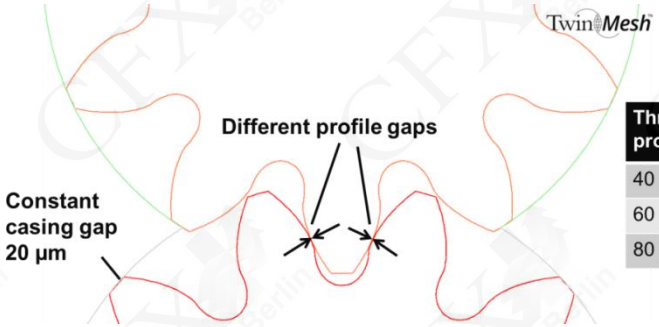
# Simulation results
## External Gear Pump

- 2D-Simulation result with mesh movement and vapour fraction



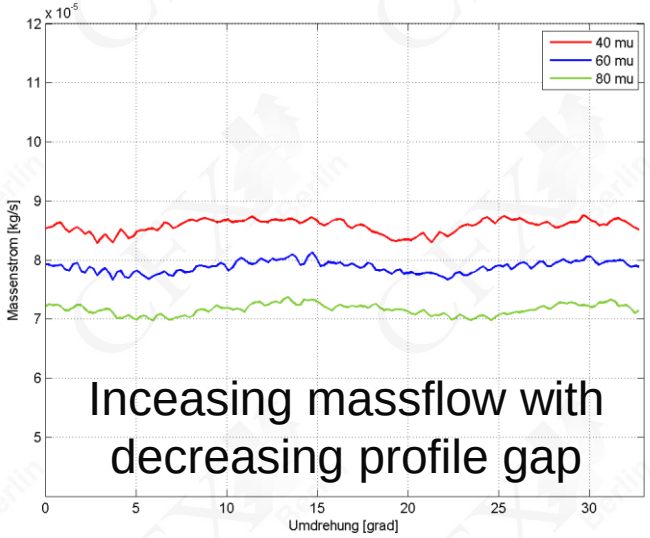
# Simulation examples
## External Gear Pump

- Possible simulation study
  - Change of the profile gap size

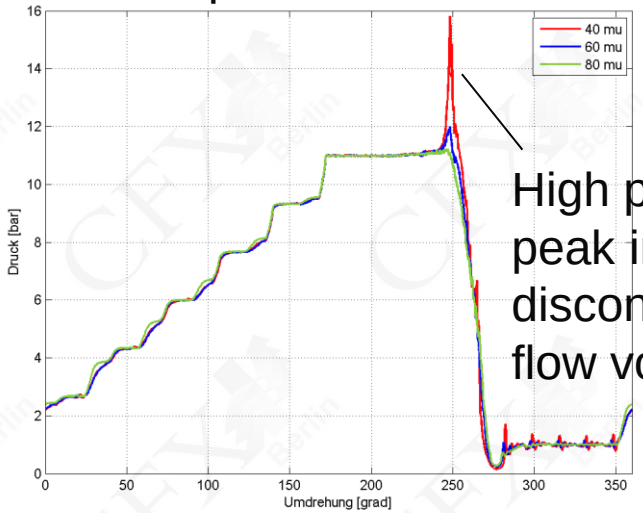



| Three different profile gaps |
|---|
| 40 µm |
| 60 µm |
| 80 µm |

Mass flow

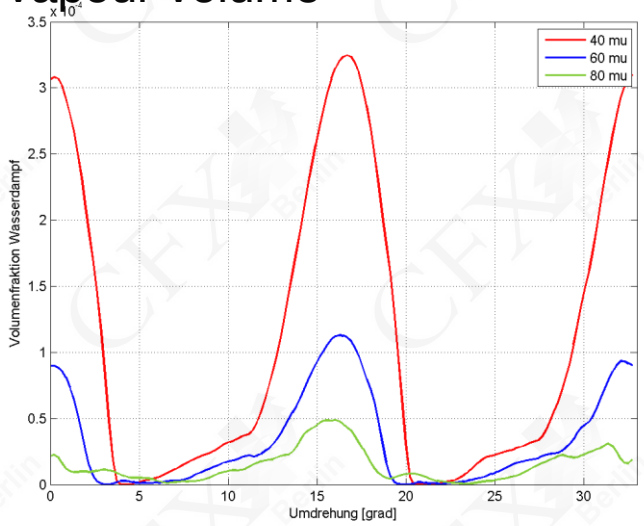



Chamber pressure

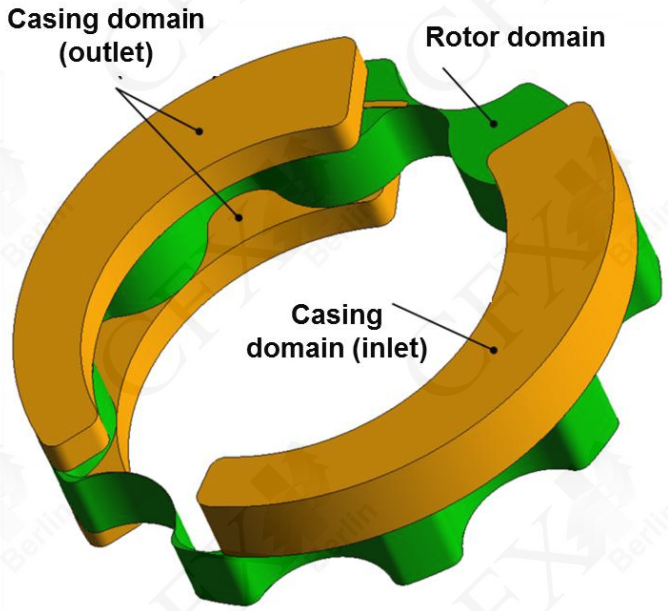


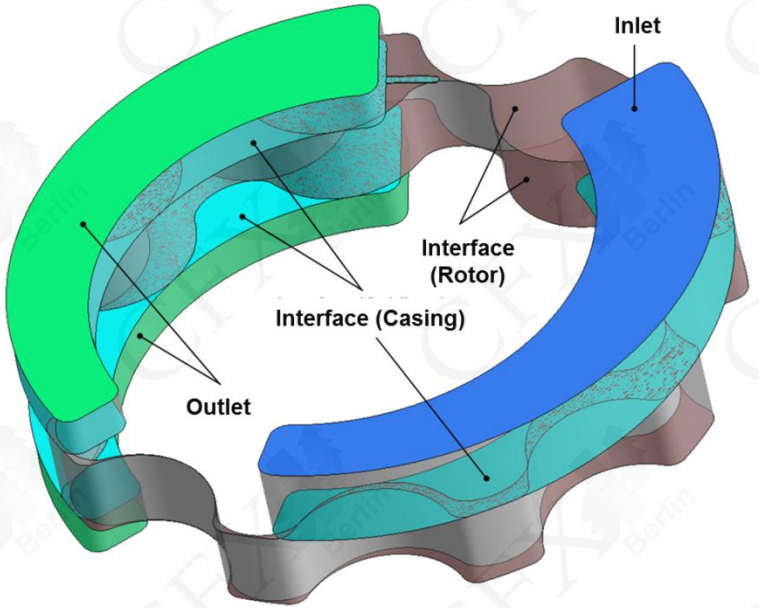

High pressure peak in disconnected flow volume

Vapour volume



- Simulation domain

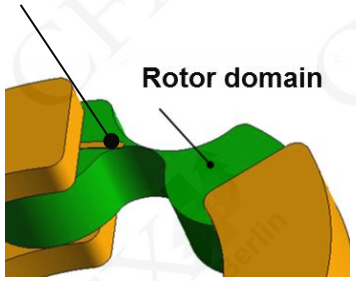


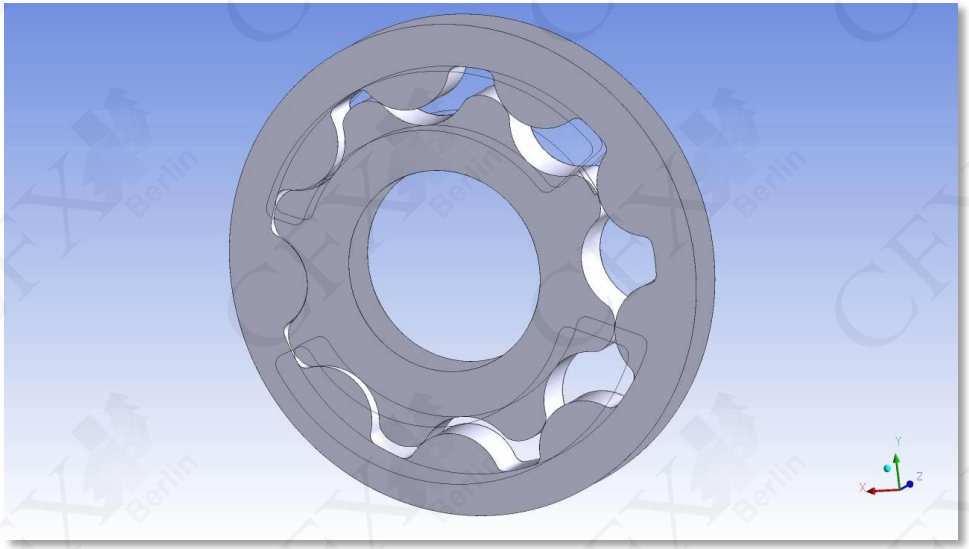



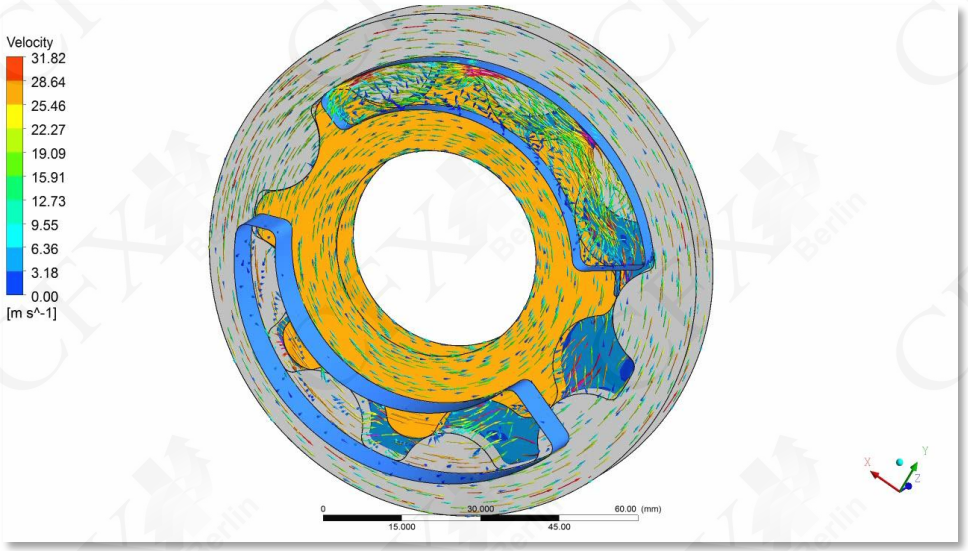

**Case without axial gaps**

# Simulation examples
## Gerotor pump

- Simulation results for case with axial gap flow
  - Time dependent information
    - Pressure pulsation
    - Flow velocities
    - Cavitation behavior
  - Average information (also transient available)
    - Mass flow
    - Torque on rotors
    - Temperature
- Possible simulation study
  - Change geometry to minimize the pressure peak

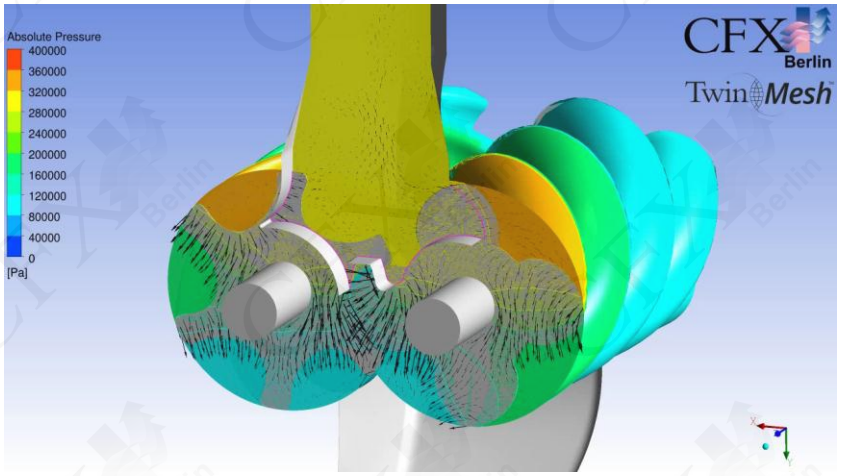


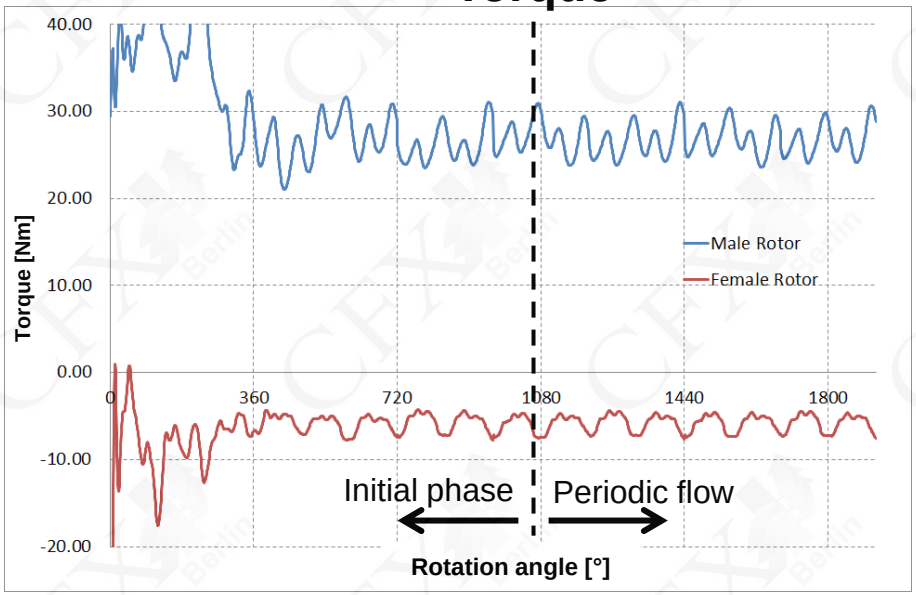



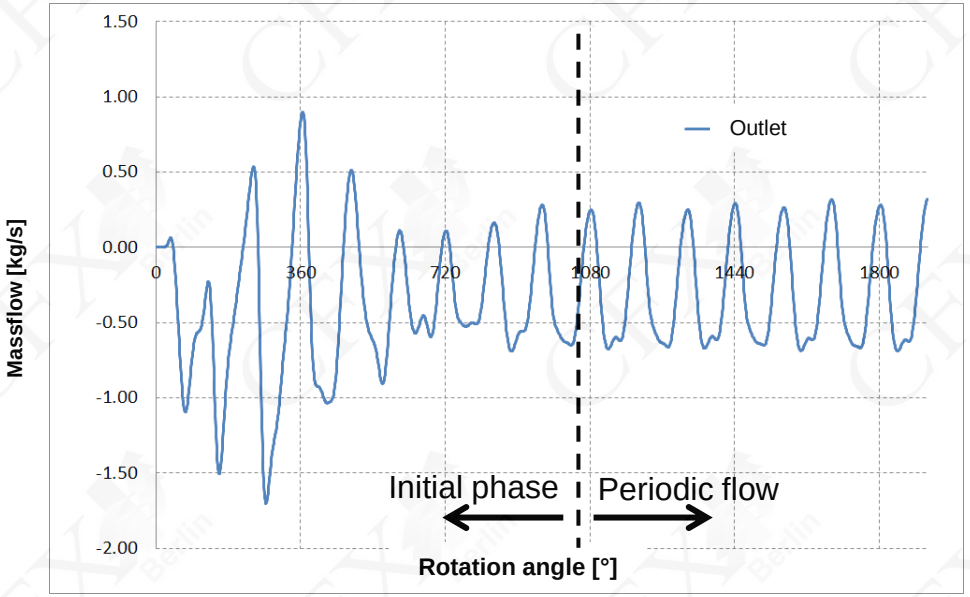


# Simulation examples
## Screw compressor

- Flow characteristics
  - Highly screwed rotors
  - Including axial gap
  - Compressible fluids
  - High flow velocities
- General analyses
  - Torque, power and massflow

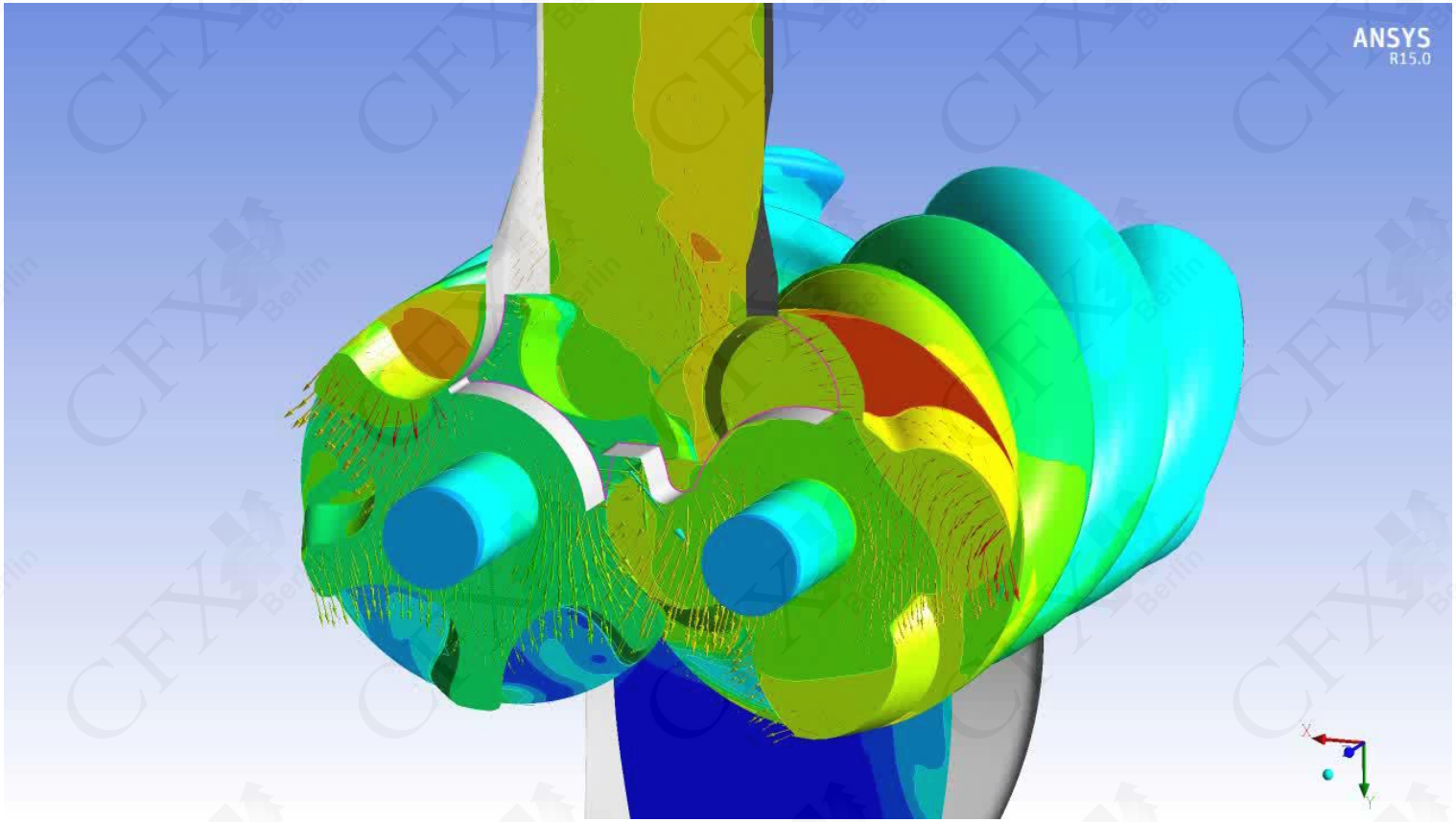


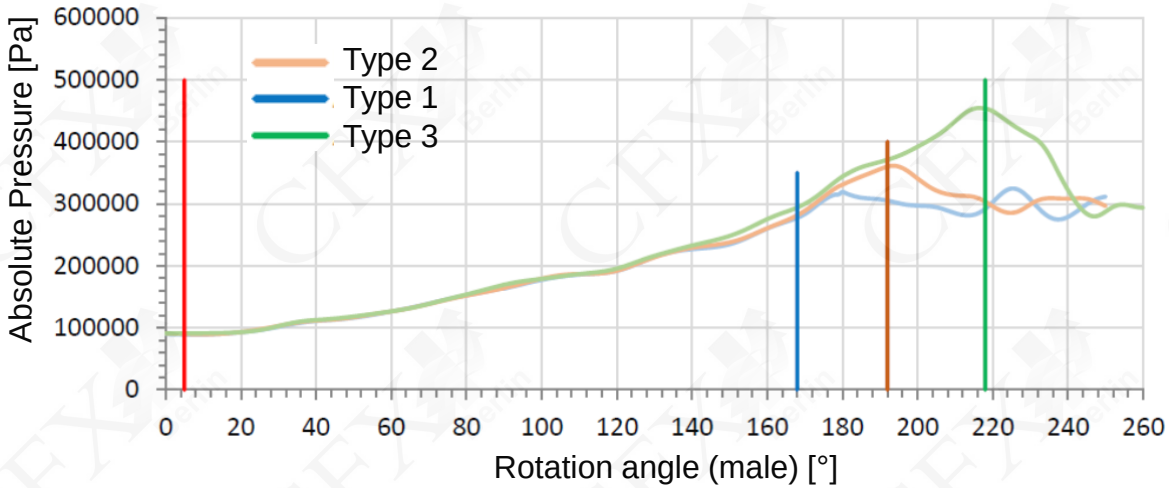



# Simulation examples
## Screw compressor

- Consideration of heat conduction in rotor solid
  - Temperature

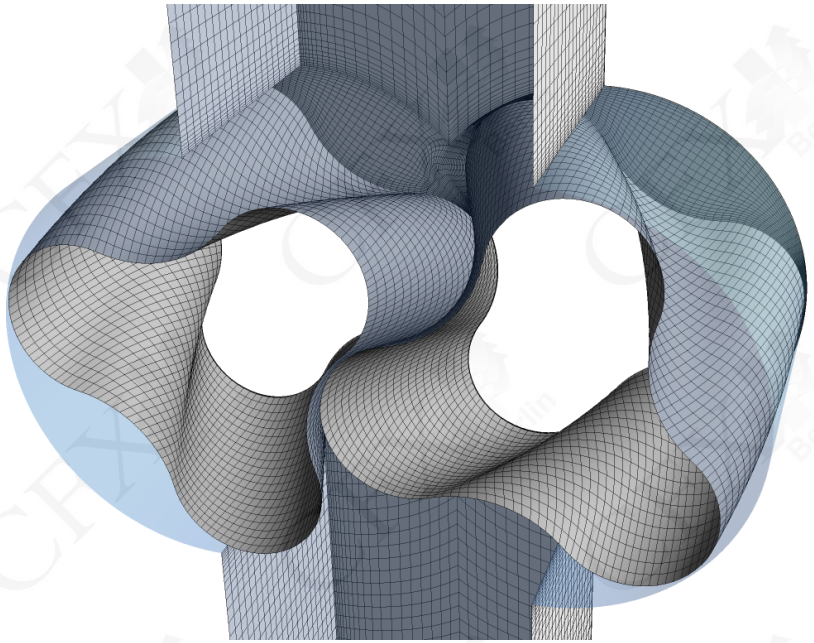


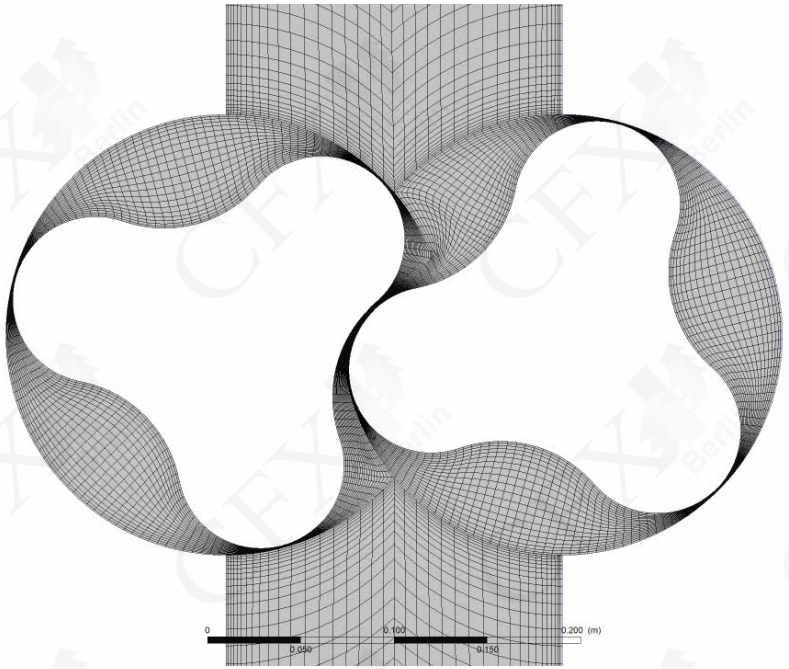
- Possible simulation study
  - Different types of discharge port geometry



# Simulation examples
## Lobe pumps

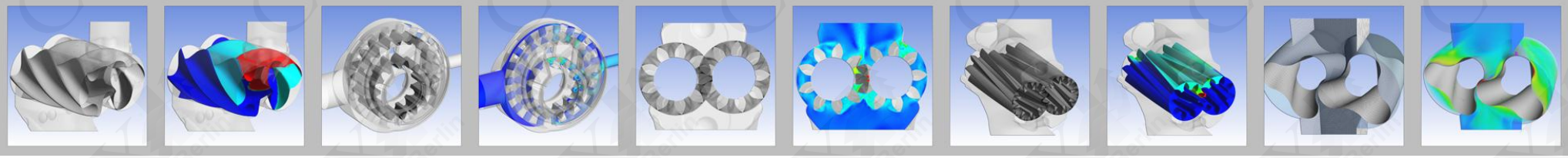
- Flow characteristics
  - Incompressible fluids with cavitation (multiphase)
  - Non-newtonian fluids even with high viscosity



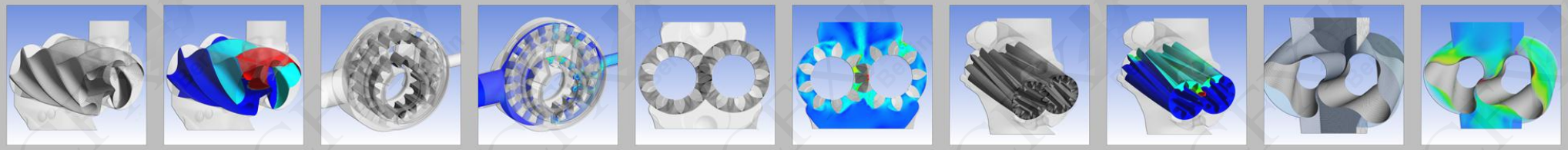


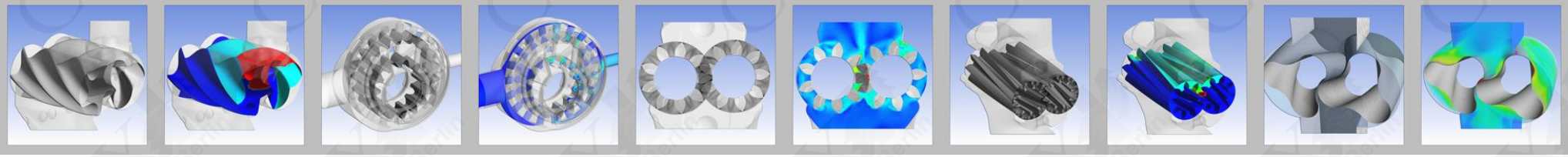
# TwinMesh™

CFX Berlin



- ✓ **Fast**: TwinMesh provides an automated workflow for structured mesh generation and CFD analysis setup of pd machines
- ✓ **Reliable**: TwinMesh delivers high quality numerical grids, avoids interpolation errors and allows the use of the full range of physics available in ANSYS CFD
- ✓ **Efficient**: TwinMesh HexMeshes allow for reasonable transient CFD model size





TwinMesh enables developers of rotary pd machines to:

- ✓ **Increase** product **durability**, **performance** and **efficiency**
  - E.g. prevention of damage by cavitation or pressure pulsation
  - E.g. optimization of flow and pressure and thermal behavior
- ✓ **Reduce development** and **manufacturing costs** by use of virtual prototyping
- ✓ **Drive** real **innovation** and **optimization** by better understanding of the machine behavior in detail
  - E.g. clearance losses, pulsation, temperature gradients and heat transfer

# Outlook

- ✓ Software developing goes forward to include more functionality to handle almost all positive displacement machine types.

- ✓ Planned extensions in future releases of TwinMesh:

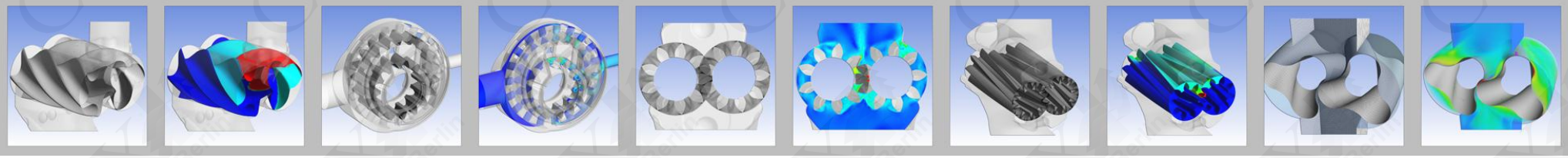
**Scroll compressor**

**Eccentric screw pump**

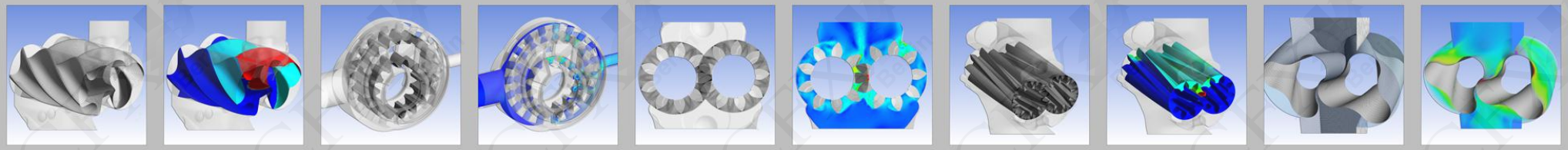
**Screw spindle pump**

TwinMesh™

CFX Berlin



CFX BERLIN

**is your partner for:**

- ✓ TwinMesh™ Software
- ✓ ANSYS Simulation Software
- ✓ Training & Support
- ✓ Consulting & Development