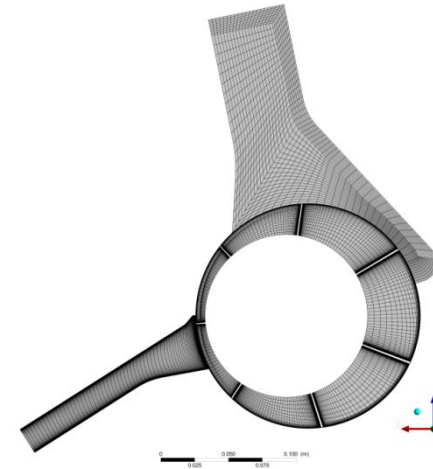# Co-Simulation
von Flownex und ANSYS CFX
am Beispiel einer Verdrängermaschine

Benoit Bosc-Bierne, Dr. Andreas Spille-Kohoff, Farai Hetze

CFX Berlin Software GmbH, Berlin

Contents

- Positive displacement compressors
- Motivation
- Coupling Flownex and ANSYS CFX
- Application cases:
    - Acoustic wave propagation
    - Vane pump
    - Screw compressor
- Summary and outlook

36. CADFEM ANSYS Simulation Conference
October 10 – 12, 2018, Congress Center Leipzig

Positive Displacement Compressors

CFX BERLIN

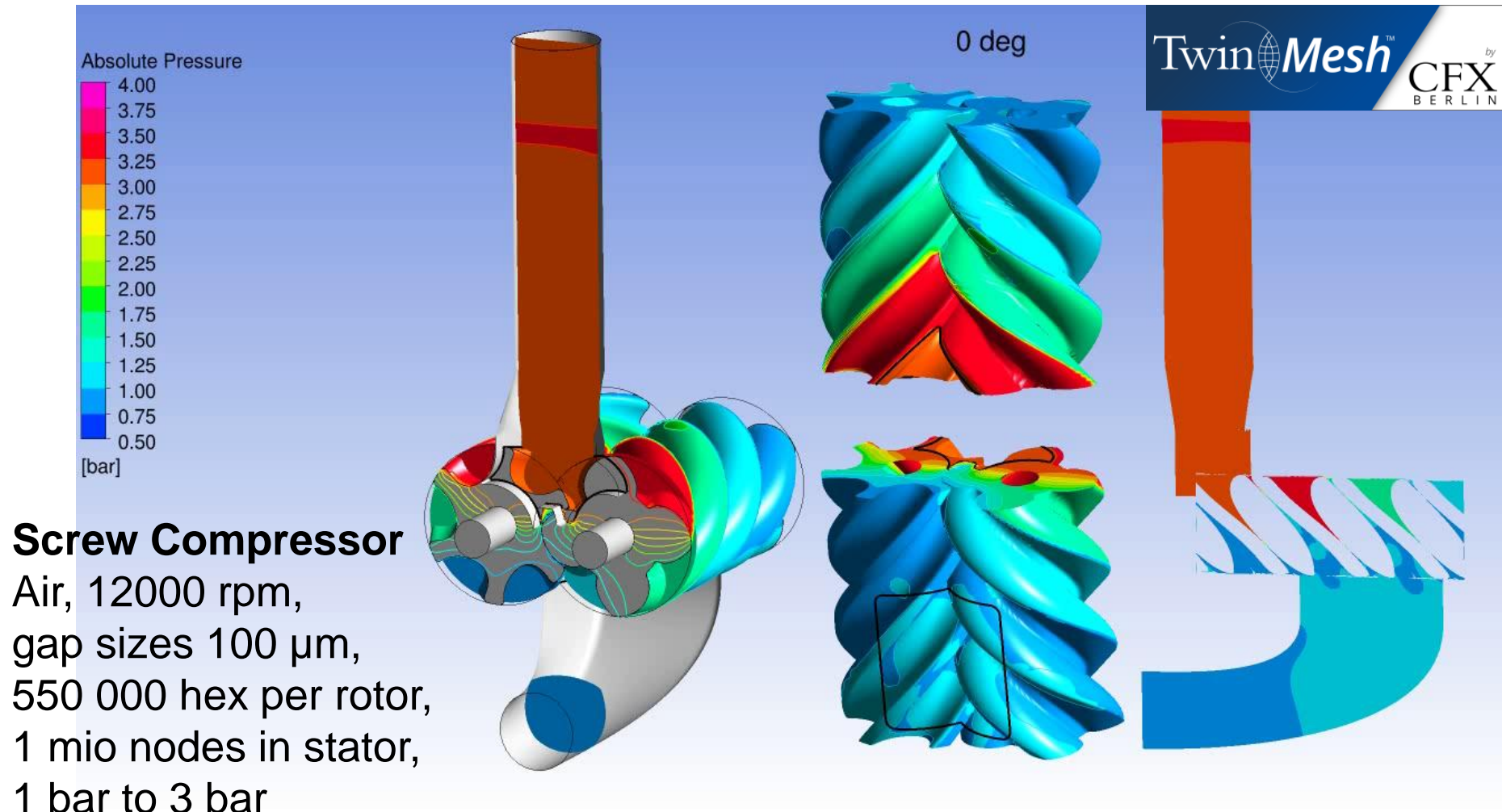3D CFD simulation of positive displacement machines at CFX Berlin:

- Established simulation process
  - with meshing, setup, simulation, and postprocessing
- Own product „TwinMesh"
  - for pre-generation of all meshes for fluid volumes in chambers
  - scripts for automated setup and reports
  - meshes are read by solver at run-time
- High numerical effort due to
  - large meshes to resolve gap flows
  - complex physics, e.g. IAPWS, CHT, MPF
  - small time step sizes to ensure convergence at high rpm
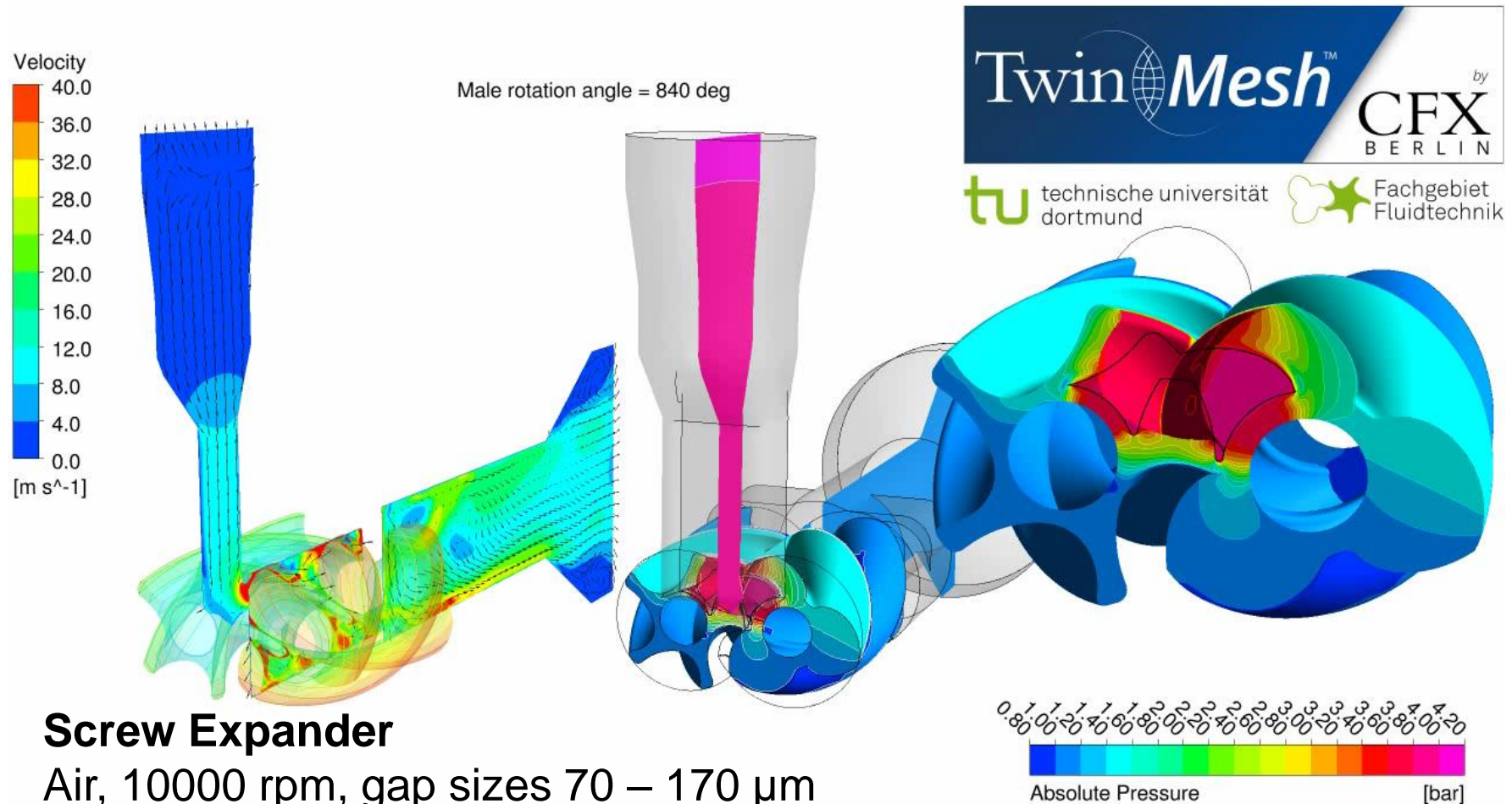 - long simulation times to reach periodic state

CAD

TwinMesh™

1. Import geometry
2. Set boundary conditions
3. Generate interfaces
4. Define mesh settings
5. Generate meshes
6. Check mesh quality
7. Export all meshes
8. Export scripts

ANSYS

1. Apply Pre script with initial mesh
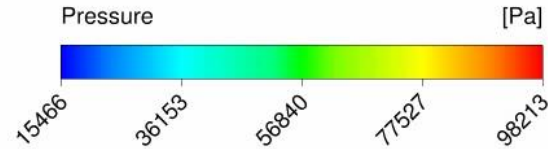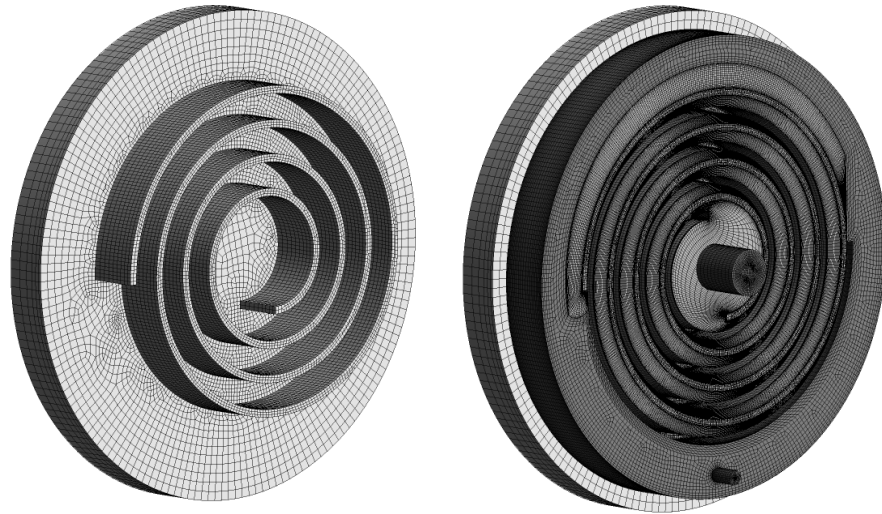2. Read further meshes at run-time

Positive Displacement Compressors


Screw Compressor
Air, 12000 rpm,
gap sizes 100 µm,
550 000 hex per rotor,
1 mio nodes in stator,
1 bar to 3 bar

36. CADFEM ANSYS Simulation Conference
October 10 – 12, 2018, Congress Center Leipzig

Positive Displacement Compressors

Screw Expander

Air, 10000 rpm, gap sizes 70 – 170 µm

1 mio hex per rotor, 480 000 cells in stator, 4 bar to 1 bar

Positive Displacement Compressors

Scroll Compressor
Air, 1700 rpm,
gap sizes 30 μm,
3 mio nodes with solids
for CHT,
20 kPa to 100 kPa

Positive Displacement Compressors
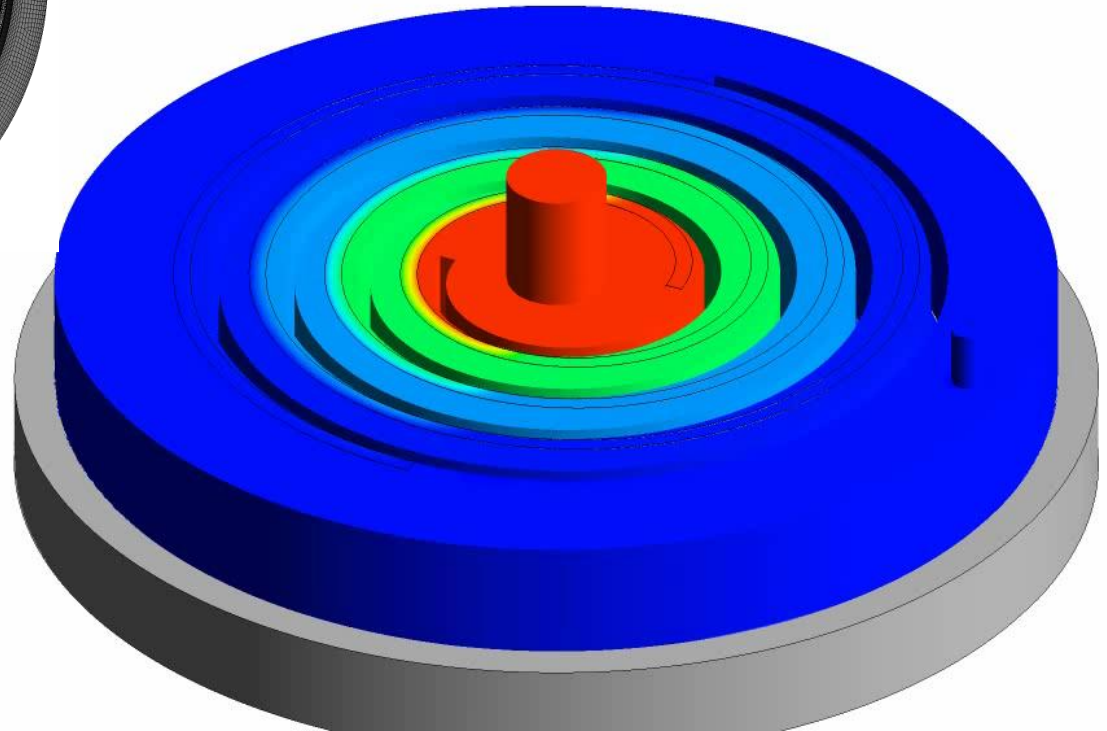
Positive displacement machines are used

- as compressors
  - for pressurized air supply for pneumatic tools
  - in process industry
  - for air conditioning systems
  - as superchargers for motors
- as expanders
  - for power generation from steam or exhaust gases
- as vacuum pumps
  - for low vacuum
  - as backing pump together with second vacuum pump

- They are always part of a system with control and feedback control mechanisms!
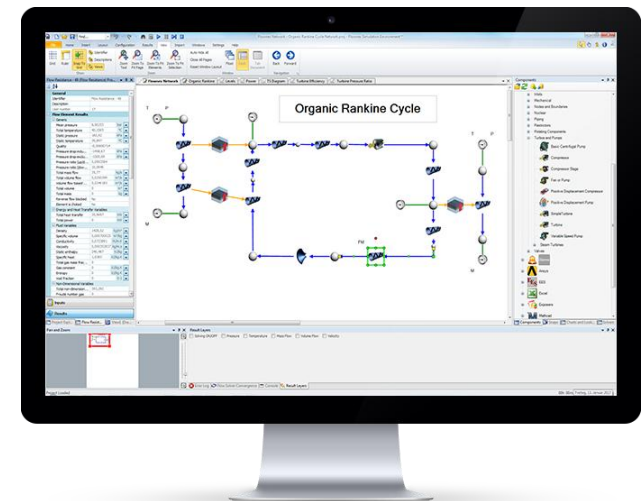
Some TwinMesh users…

- 3D CFD analysis of compressors is time consuming due to
  - Fine meshes with a lot of elements
  - Complex flow phenomena
  - Transient simulations with small time step sizes
- Thus, 3D CFD analysis should focus on the component itself
- But:
  - Artificial boundaries (pressure openings) are necessary
    ➔ unknown boundary conditions, unphysical interaction with boundaries
  - Interaction with system (pipes, storage vessels, valves, consumer loads, failure / start-up scenario) requires inclusion of more components into 3D CFD
    ➔ larger meshes with longer simulation times
- Alternative: Co-simulation of 3D CFD with 1D CFD

1D network model of suction side ↔ **Component 3D CFD** ↔ **1D network model of pressure side**

Flownex solves 1D-CFD system-level network for mass and heat transport:

- Conservation of mass, momentum, and energy

- Incompressible and compressible fluids

- Gases, gas and gas-liquid mixtures, liquids, slurries, non-newtonian liquids

- Single and multi-phase flows

- Steady-state and transient

- Huge pool of components like pipes, vessels, junctions, valves, orifices, pumps with characteristic curves

- New components with special properties can be defined easily

- Integrated coupling e.g. to ANSYS, EXCEL

36. CADFEM ANSYS Simulation Conference
October 10 – 12, 2018, Congress Center Leipzig

Coupling Flownex and ANSYS CFX

Flownex has generic, file-based interface to ANSYS CFX:

- User selects input and output variables (may depend on flow direction)

- Flownex starts ANSYS CFX solver

- After each time step, Flownex writes output variables into file, and waits for input data from ANSYS CFX

ANSYS CFX uses User Fortran routines:

- Read Flownex data at start of each time step and set as boundary condition

- Simulate time step (with inner iterations) → explicit coupling
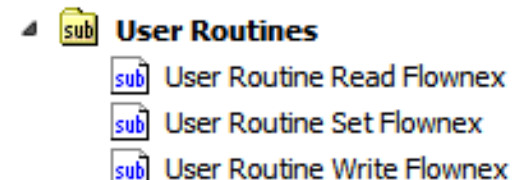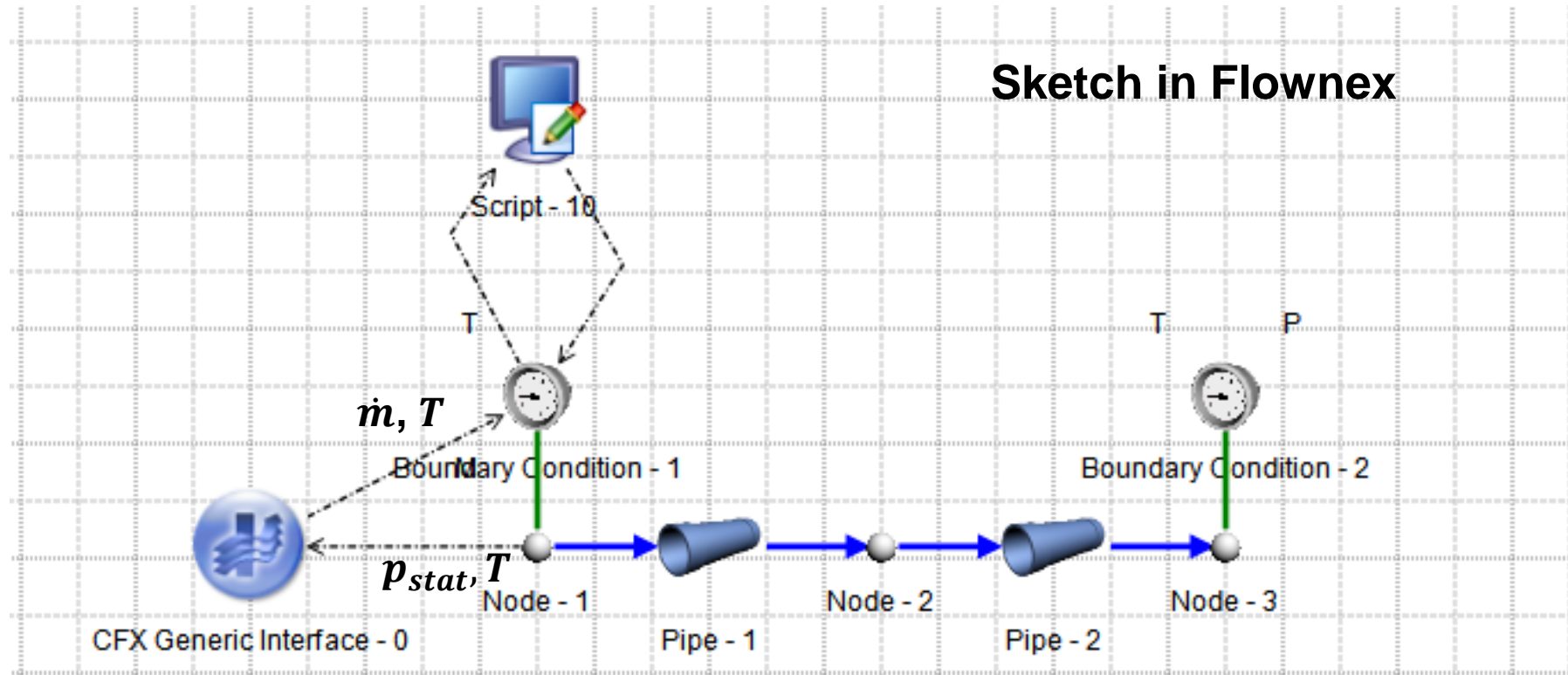
- Write input data for Flownex

CFX Generic Interface - 1

Input into Flownex

| Mass Flow Rate (inlet/outlet) |
| Normal Speed (inlet/outlet) |
| Total Pressure (inlet only) |
| Static Pressure (inlet/outlet) |
| Average Static Pressure (outlet only) |
| Stat. Frame Tot. Press. (inlet & turbo mode only) |
| Total Temperature (inlet only) |
| Static Temperature (inlet only) |
| Stat. Frame Total Temp. (inlet & turbo mode only) |

Output to ANSYS CFX

| Mass Flow |
| Total Pressure |
| Absolute Pressure |
| Temperature |

ANSYS 19.2 RELEASE

User Routines
- User Routine Read Flownex
- User Routine Set Flownex
- User Routine Write Flownex

36. CADFEM ANSYS Simulation Conference
October 10 – 12, 2018, Congress Center Leipzig

Acoustic wave propagation

Sketch in Flownex

- Geometry
  - 1D pipe in ANSYS CFX with L = 0.20 m with 200 elements
  - Pipe - 1 with L = 0.50 m and 50 increments
  - Pipe - 2 with L = 0.15 m and 15 increments
  - Fixed pressure boundary at Node-3

  0.85 m total length

Acoustic wave propagation
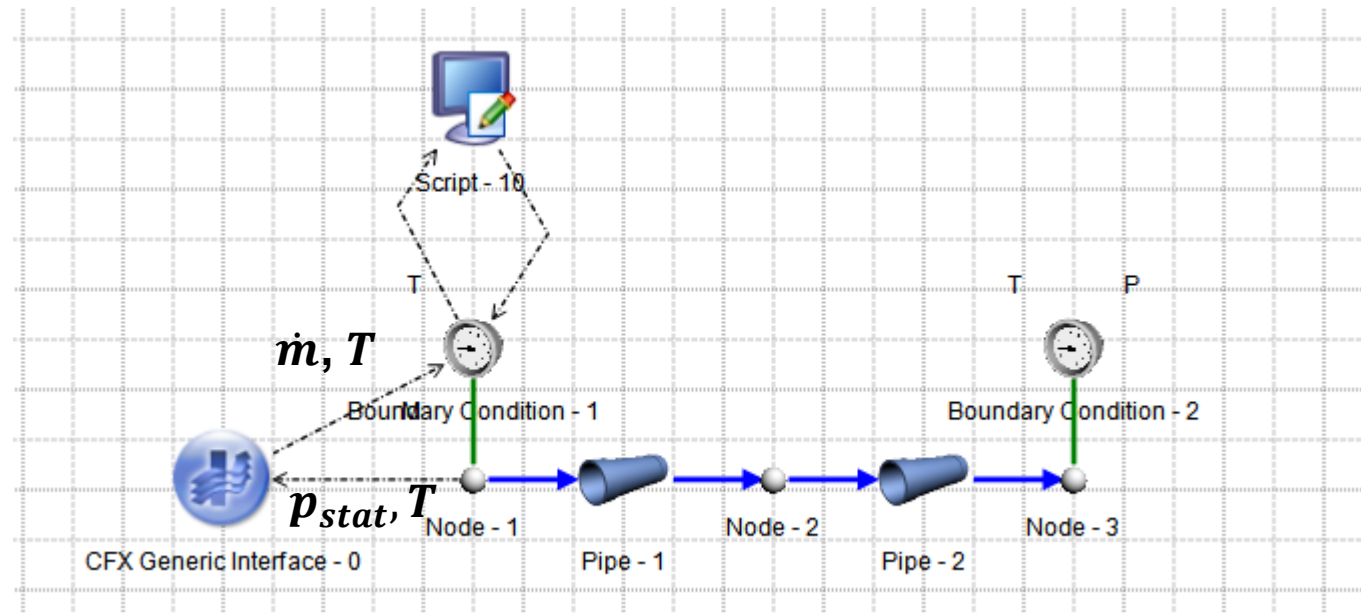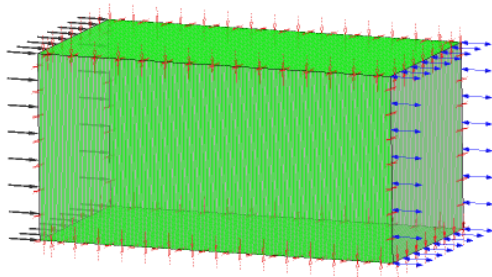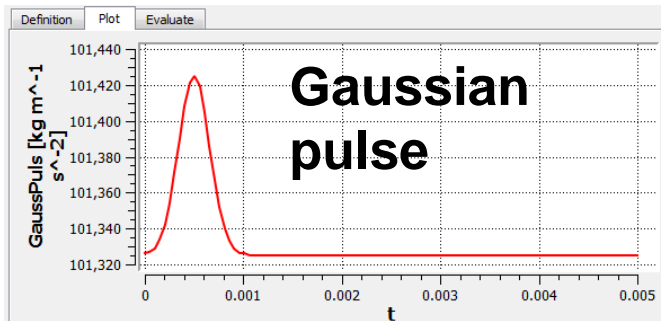
- Compressible transient simulation in Flownex and ANSYS CFX with air
- Gaussian pressure pulse specified in ANSYS CFX at inlet
- ANSYS CFX gives mass flow and average temperature to Flownex
- Flownex gives pressure and temperature to ANSYS CFX
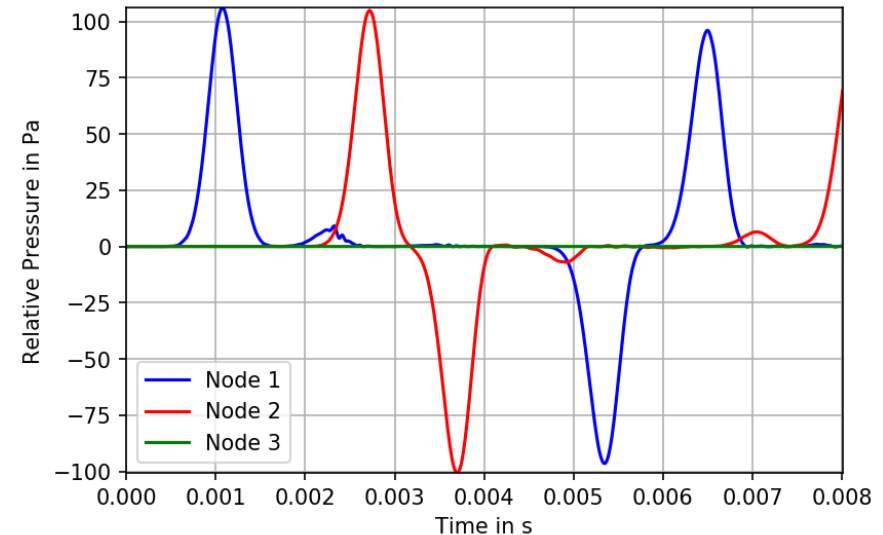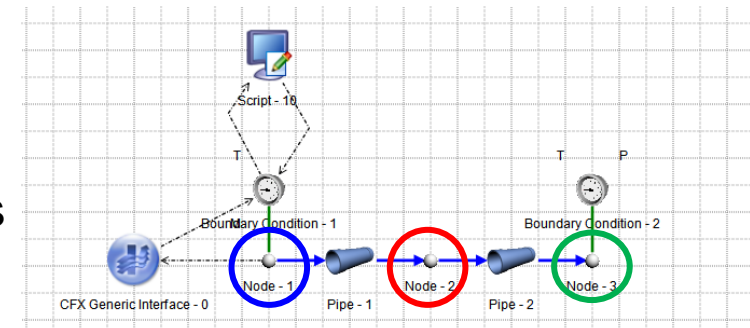- 1600 time steps à 5 µs → 8 ms simulation time → 2.8 m travel distance

Acoustic wave propagation
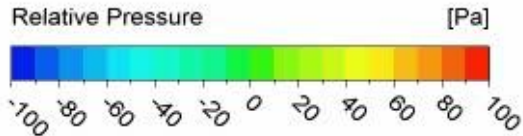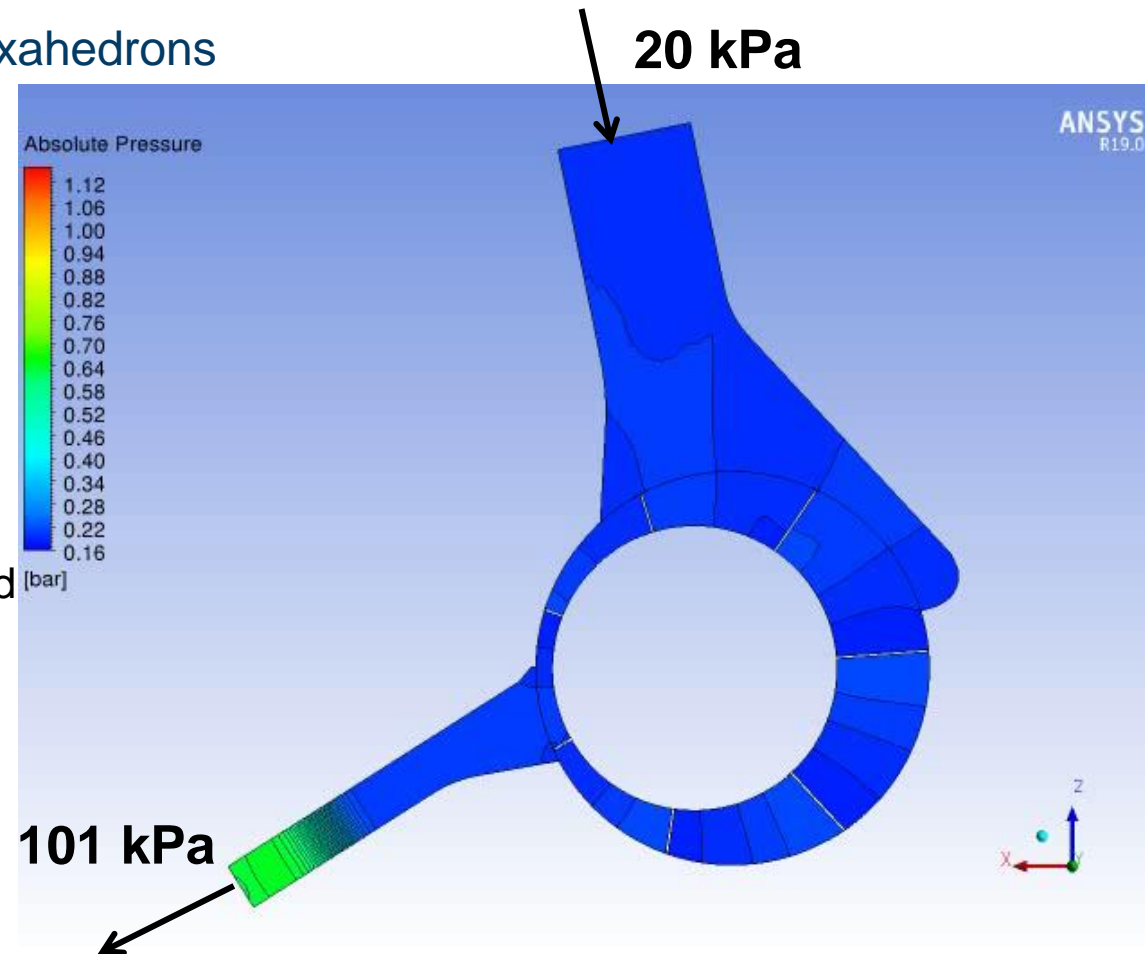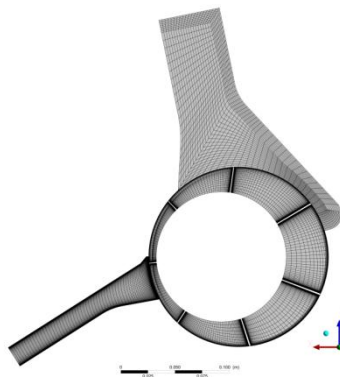

ANSYS CFX

Flownex

- Pressure pulse leaves ANSYS CFX domain and enters Flownex at 1 ms
- Travels towards Flownex' boundary, is reflected at Node-3 and travels back
- Enters ANSYS CFX domain at 5 ms, travels towards inlet, is reflected and travels again to right
- Enters Flownex at 6 ms…
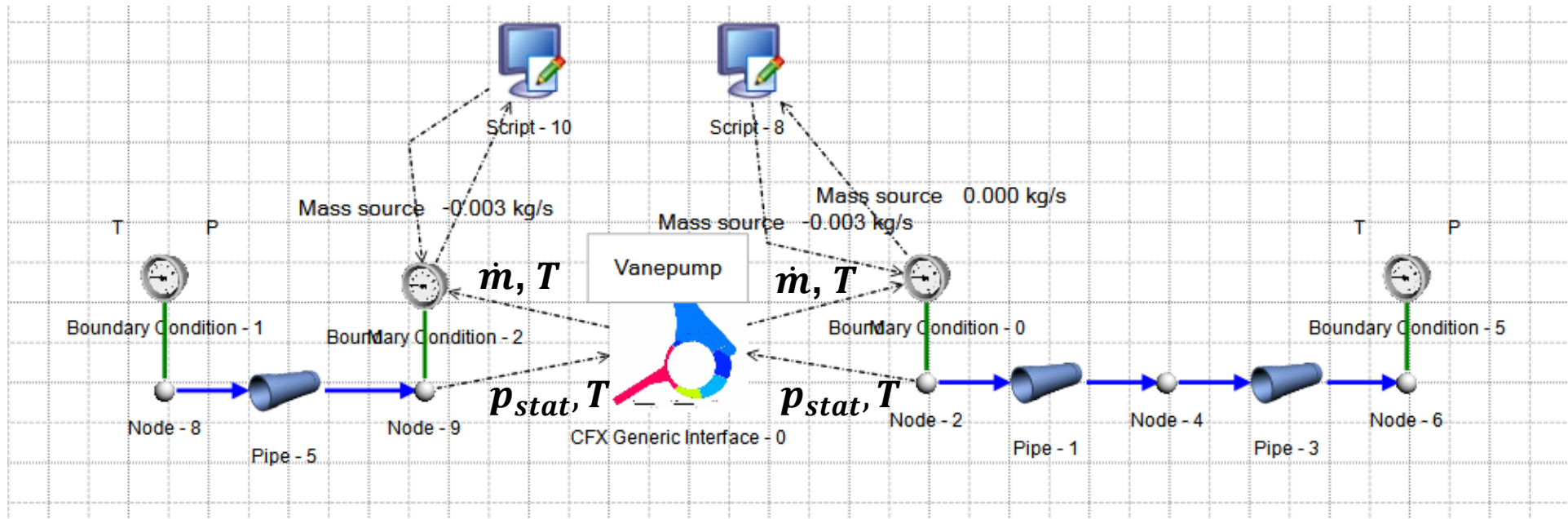
Vane pump

Vane pump model in ANSYS CFX:

- Quasi-2D mesh with 45 000 hexahedrons

- Fluid

  - Air as ideal gas

- SST turbulence model

- Boundary conditions:

  - Rotational speed: 2380 rpm

  - Inlet at $p_{in} = 20 \text{ kPa}$

  - Outlet at $p_{out} = 101.325 \text{ kPa}$

  - Openings specified as standard (reflective) boundaries

20 kPa

101 kPa

Absolute Pressure

1.12
1.06
1.00
0.94
0.88
0.82
0.76
0.70
0.64
0.58
0.52
0.46
0.40
0.34
0.28
0.22
0.16

[bar]

ANSYS
R19.0

36. CADFEM ANSYS Simulation Conference
October 10 – 12, 2018, Congress Center Leipzig

Vane pump

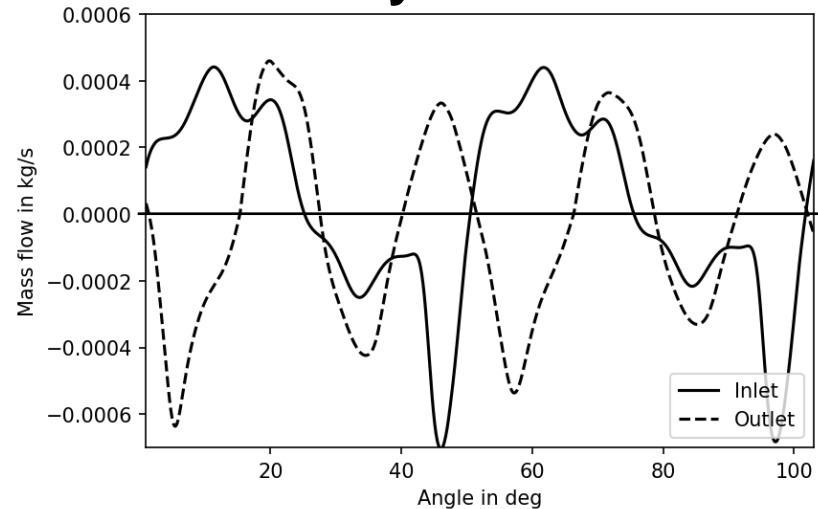Flownex system with ANSYS CFX co-simulation:

- Pipes added as 1D models:
  - 0.1 m at suction side of vane pump (Pipe-5)
  - 0.5 m and 0.15 m at pressure side of vane pump (Pipe-1 and -3)
- Pressure boundary conditions set at Flownex boundaries (Node-8 and -6)

Vane pump

Comparison of results:

- Periodical mass flow for 51.4 deg rotation angle

- Results show high pulsation amplitudes for uncoupled simulation (standing waves) at inlet and outlet

- Coupled simulation has smaller pulsation amplitudes in inlet and outlet mass flow rate, shape of pulsation also changes

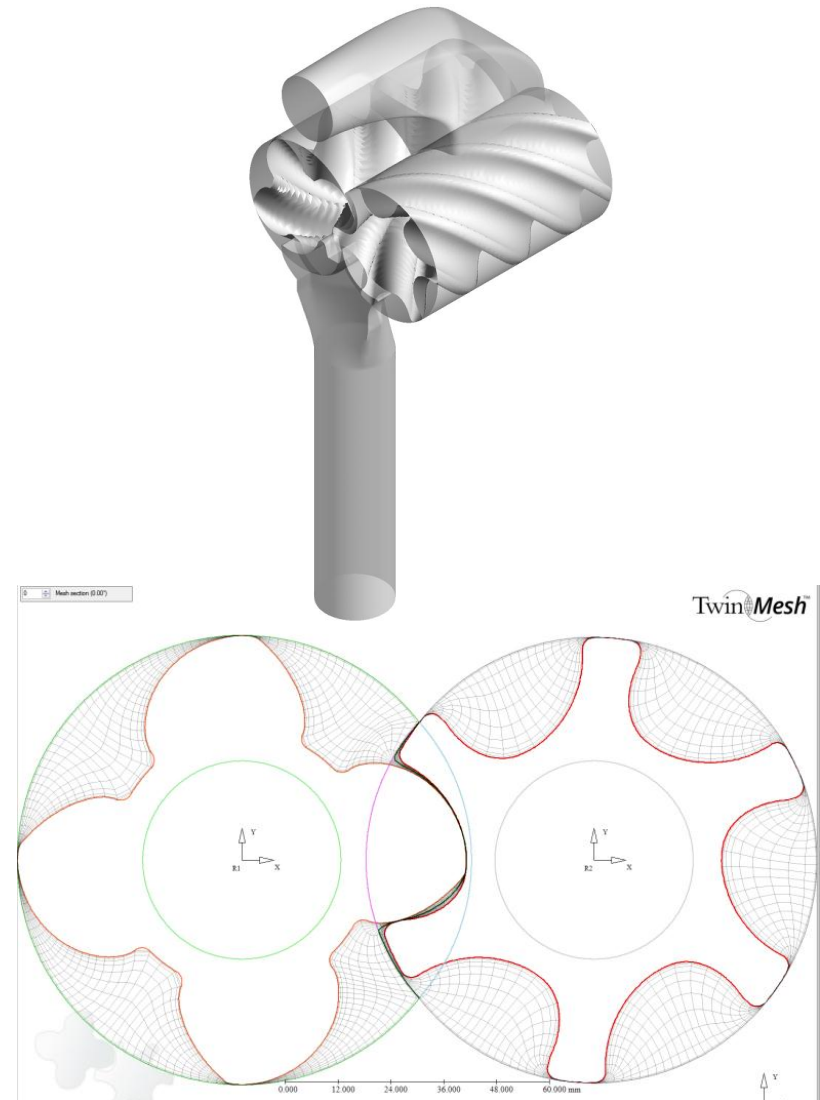- Co-simulation can reproduce real system behaviour with full interaction

only ANSYS CFX

co-simulation with Flownex

36. CADFEM ANSYS Simulati
October 10 – 12, 2018, Congres
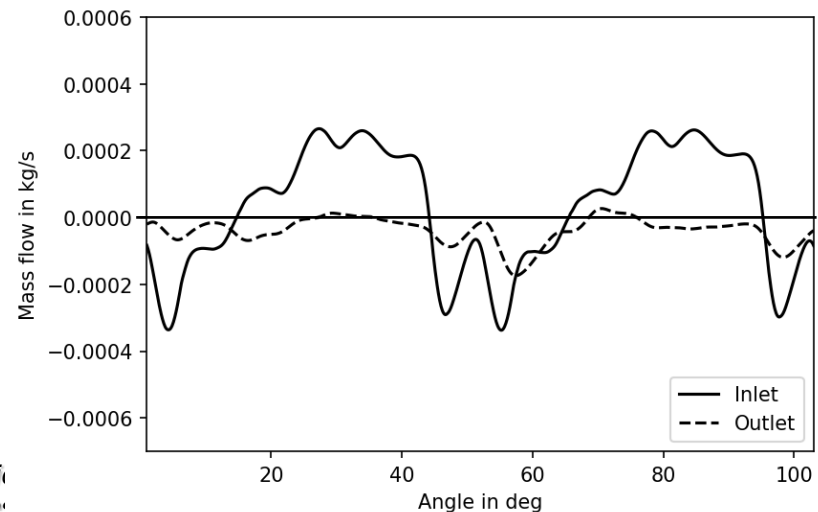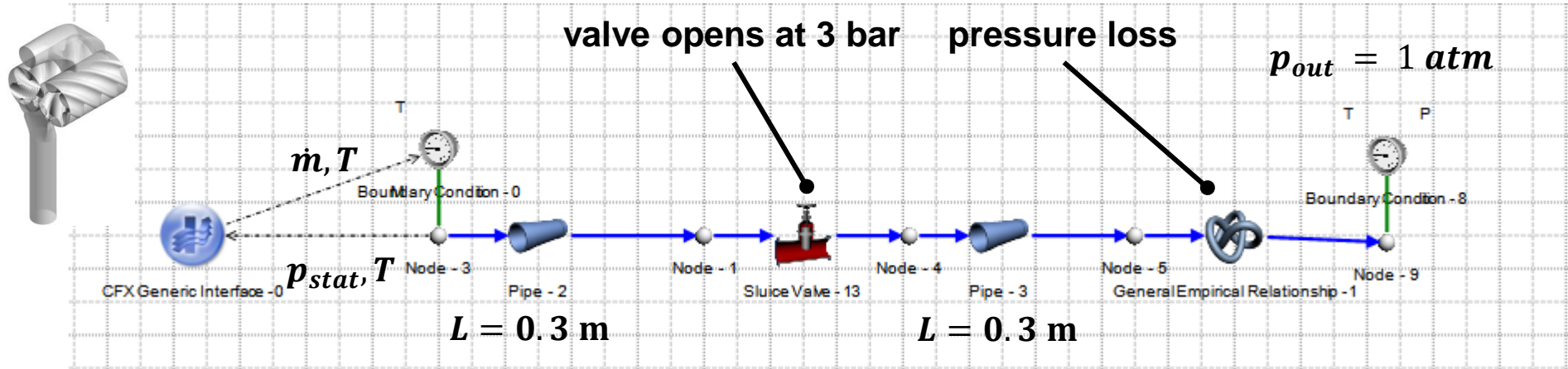
Screw compressor

Screw compressor in ANSYS CFX:

- Unstructured meshes for stationary domains created with ANSYS Meshing

- Structured meshes for rotating domains created with TwinMesh for each 5°
  - 10 radial, 166 circumferential, 50 axial = 83 000 hexahedrons per rotor

- Fluid: Air as ideal gas

- SST turbulence model

- Boundary conditions:
  - Rotational speed: 12333 rpm
  - Inlet at $p_{in} = 1$ bar, $\mathrm{T} = 20°\mathrm{C}$
  - Outlet coupled to Flownex

- Time step size 68 µs (for 5° increment)

- approx. 2 h simulation time for one revolution on 4 cores

36. CADFEM ANSYS Simulation Conference
October 10 – 12, 2018, Congress Center Leipzig

Flownex system with ANSYS CFX co-simulation:

valve opens at 3 bar   pressure loss

$p_{out} = 1\,atm$

$\dot{m}, T$

Boundary Condition - 0

p_{stat}, T Node - 3

CFX Generic Interface - 0

$L = 0.3\ \text{m}$

Node - 1

Pipe - 2

Sluice Valve - 13

Node - 4

Pipe - 3

$L = 0.3\ \text{m}$

Node - 5

General Empirical Relationship - 1

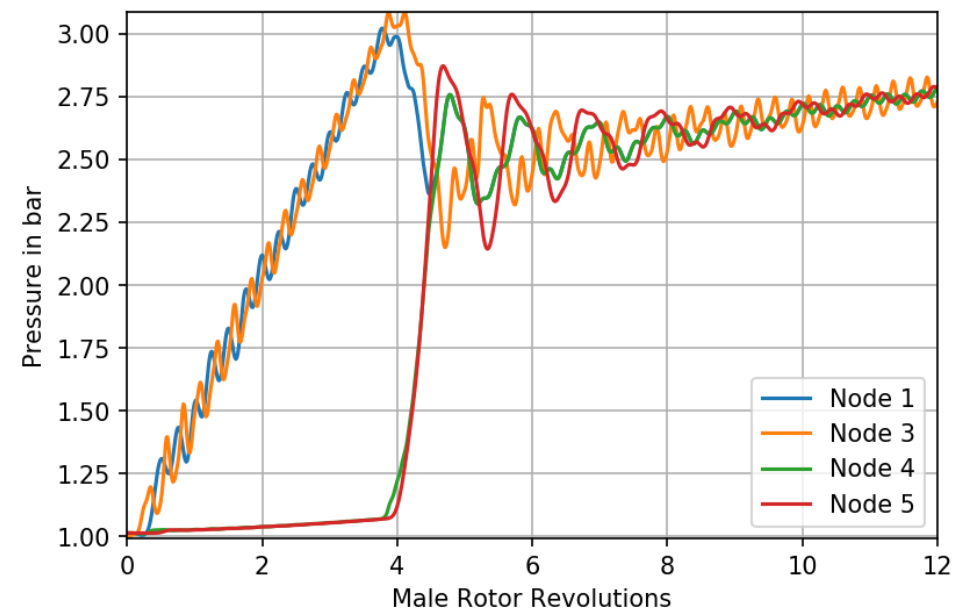Boundary Condition - 8

Node - 9

- Flownex coupled to outlet of screw compressor
- Whole system initialised at 1 bar with (almost) closed valve
- Valve opens at 3 bar (design pressure ratio 1:3 for screw compressor)

Screw compressor

Results for Flownex system with ANSYS CFX co-simulation:

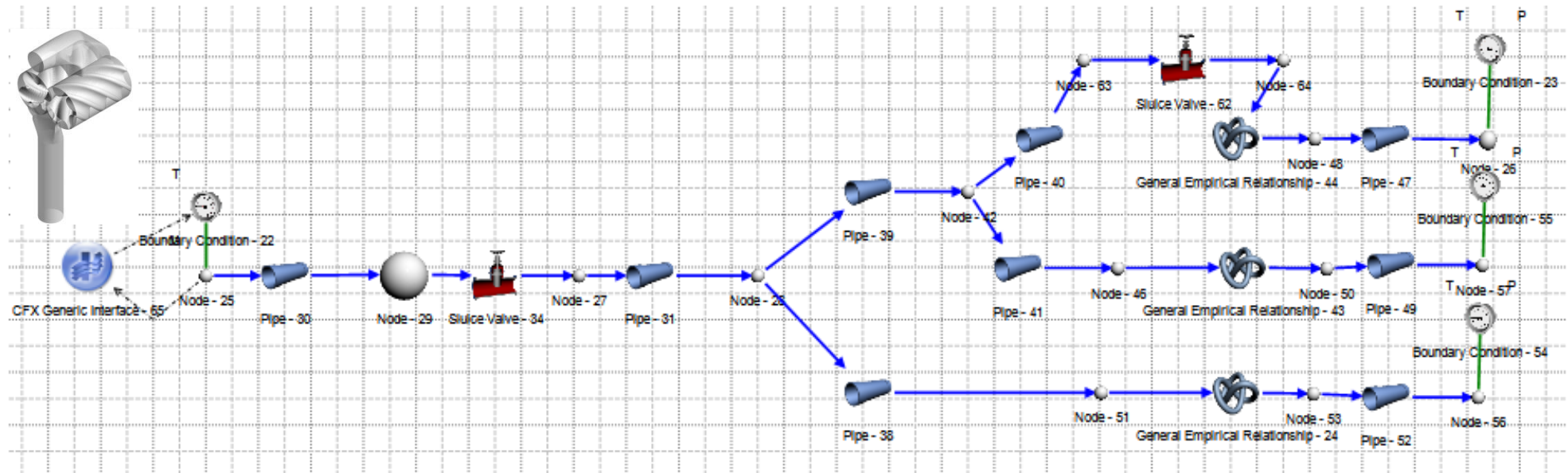Mass flow at ANSYS CFX boundaries:
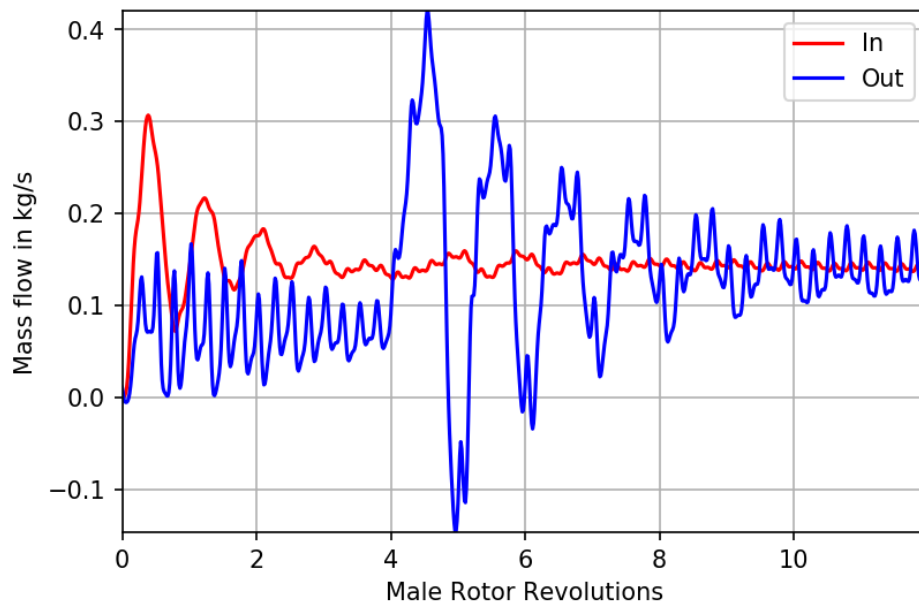
Pressure at Flownex nodes:

- ANSYS CFX outlet region and pipe-2 are pressurized up to 4 revolutions

- Valve opens and air fills pipe-3

- Pressure waves travel through pipes and cause mass flow pulsations

36. CADFEM ANSYS Simulation Conference
October 10 – 12, 2018, Congress Center Leipzig

Screw compressor

More complex Flownex system with ANSYS CFX co-simulation:

- Air storage vessel at Node-29 with volume 4 l

- Junctions to pipe systems towards different consumers

- Additional valves to switch consumers and storages

- Simulation time is mainly determined by ANSYS CFX and necessary rotor revolutions (4 l → 15 g @ 3 bar → 0.1 s @ 0.15 kg/s → 20 rev)

Summary and outlook

- 3D CFD simulation of positive displacement machines
  - Established tool for design and optimization
  - Fine meshes, complex physics, fast rotation, transient behaviour may require long simulation times
  - 3D CFD focuses on component with artificial boundaries
- 1D CFD allows fast simulation of attached fluidic networks with control mechanisms
- Co-simulation of 3D and 1D CFD
  - Takes interactions between systems into account
  - Considers control and feedback control mechanisms
  - Simulation time mainly determined by 3D CFD

- ➤ Conditional co-simulation switches between:
  - PD machine as 1D component with performance curves in standard situations → fast simulation
  - PD machine as 3D component via co-simulation when interaction is important → accurate results