CFX BERLIN

Co-simulation of 3D CFD model for positive displacement compressor and 1D CFD model of connected system

Dr. Andreas Spille-Kohoff, Rainer Andres, Farai Hetze

CFX Berlin Software GmbH, Berlin, Germany

9th International Conference on Compressor and Refrigeration

July 10th-12th, 2019

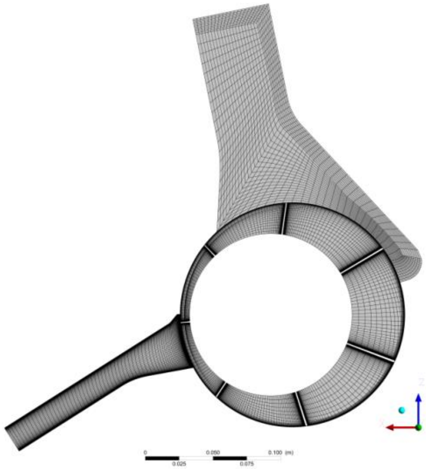
2019

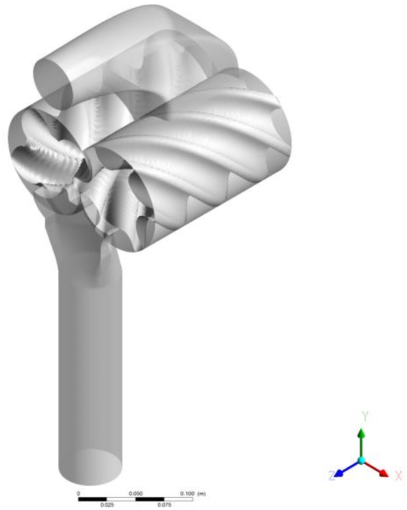
西安交通大學

XI'AN JIAOTONG UNIVERSITY

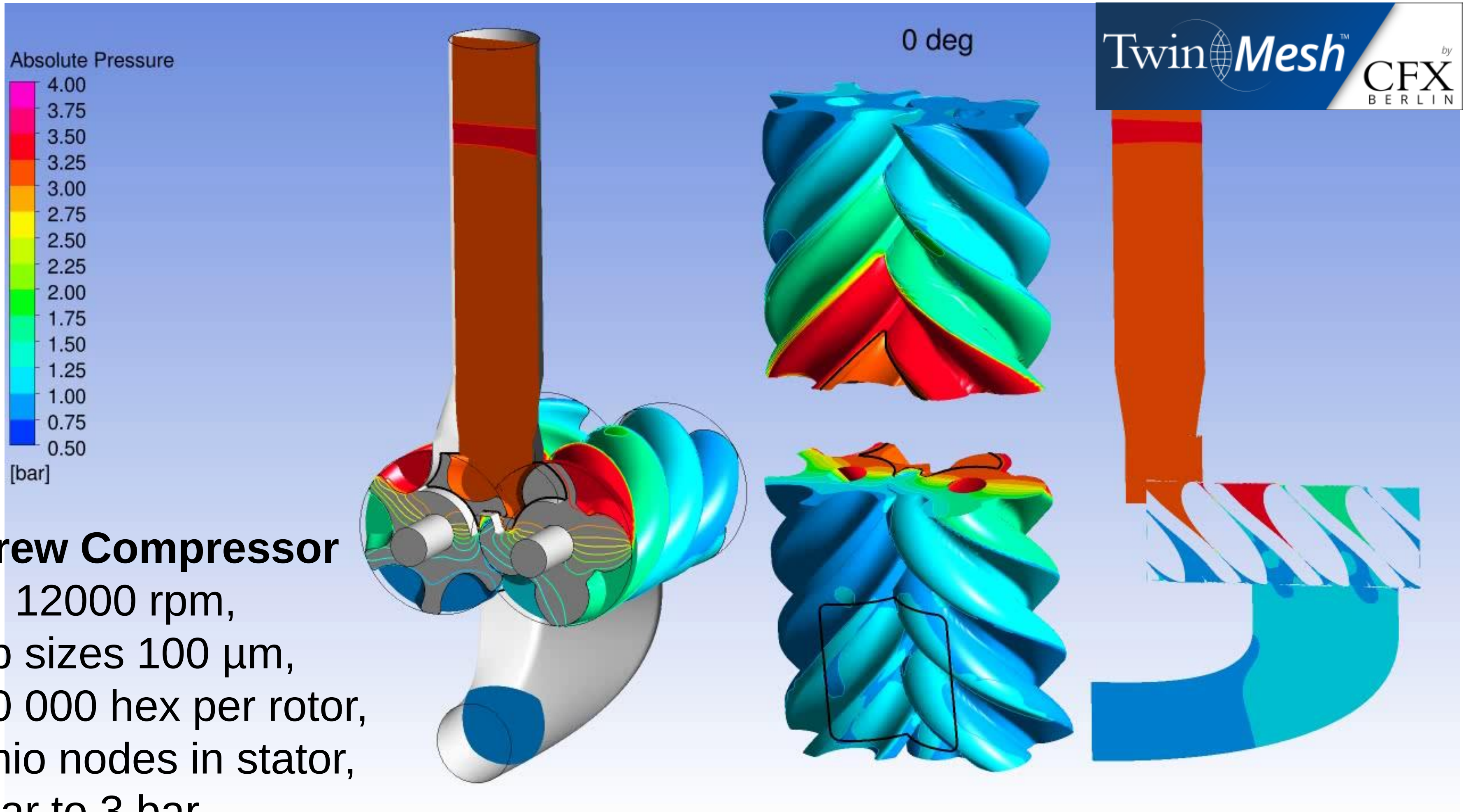
Contents

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





Motivation



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

Motivation

- 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**

Motivation

CFX BERLIN

1D network model ⟷ **Component 3D CFD**

FLOWNEX® SIMULATION ENVIRONMENT

ANSYS®

- Pool of components like pipes, vessels, junctions, valves, orifices, pumps with characteristic curves
- O(1) to O(10^2) nodes per component
- Fast simulation of fluid behavior in network

- Geometry of component fully resolved by mesh and physics of flow simulated or modeled
- O(10^5) to O(10^7) nodes per component
- Accurate simulation of fluid behavior in component

Conservation of mass, momentum, and energy

Coupling Flownex and ANSYS CFX

CFX BERLIN

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

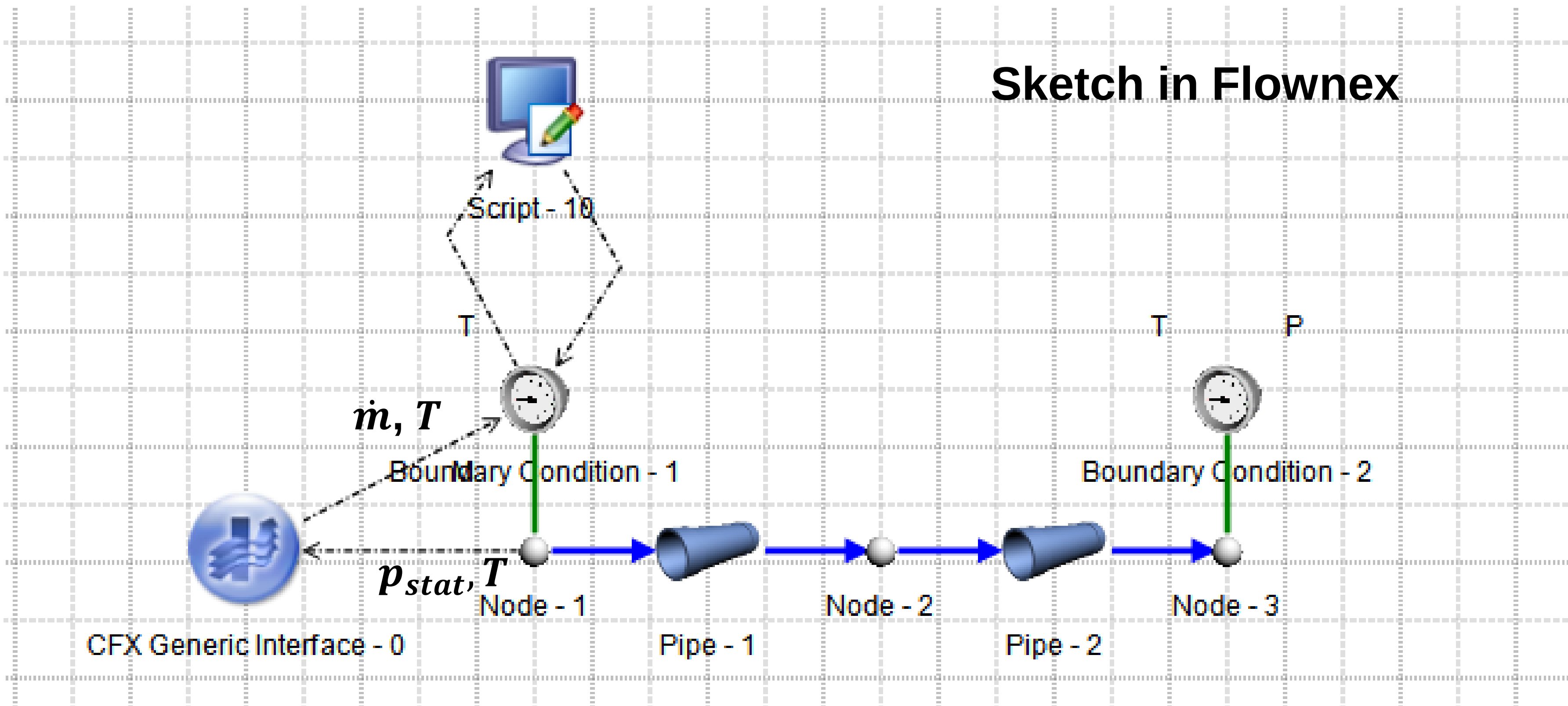
Input into Flownex	Output to ANSYS CFX
Mass Flow Rate (inlet/outlet)	Mass Flow
Normal Speed (inlet/outlet)	Total Pressure
Total Pressure (inlet only)	Absolute Pressure
Static Pressure (inlet/outlet)	Temperature
Average Static Pressure (outlet only)	
Stat. Frame Tot. Press. (inlet & turbo mode only)	
Total Temperature (inlet only)	
Static Temperature (inlet only)	
Stat. Frame Total Temp. (inlet & turbo mode only)	

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

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

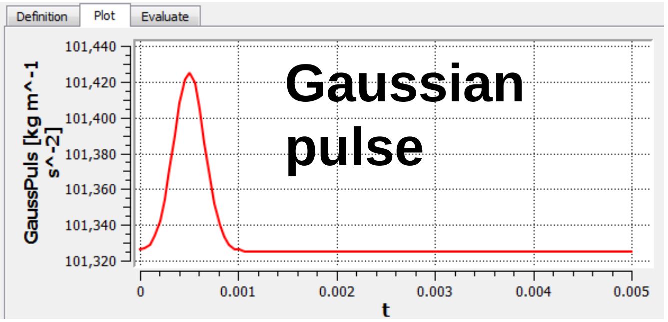
Acoustic Wave Propagation

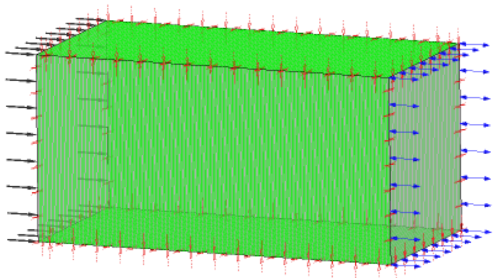


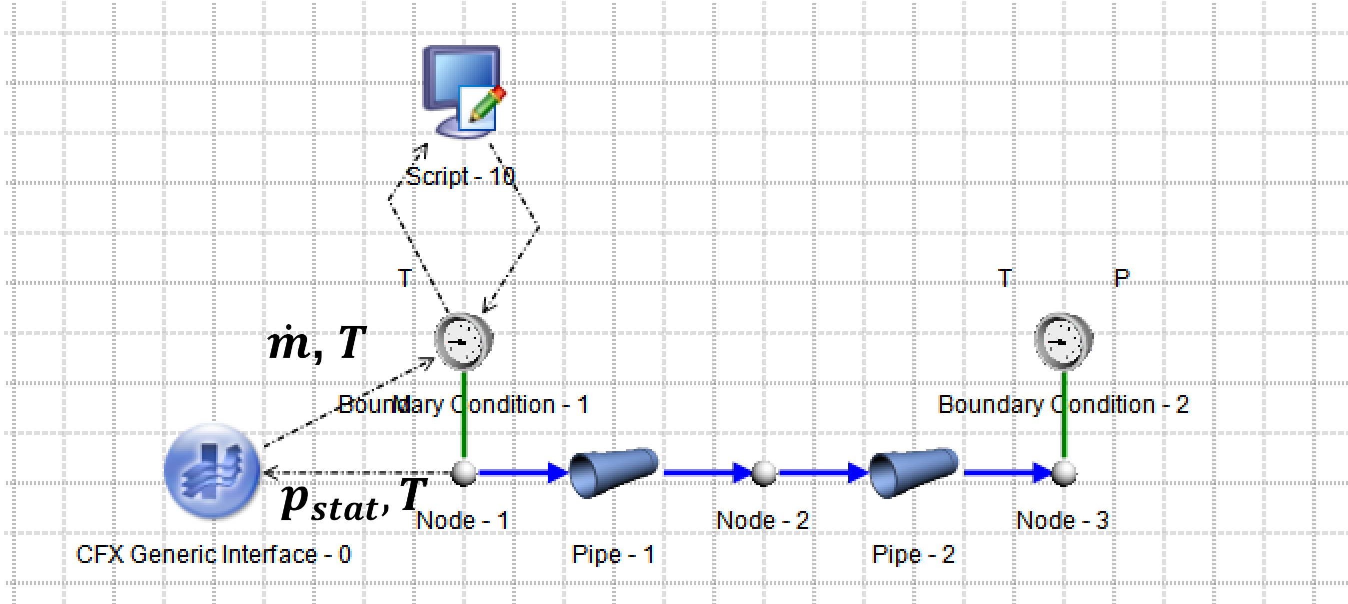
- Geometry
  - 1D pipe in ANSYS CFX with L = 0.20 m with 200 elements
  - Pipe - 1 with L = 0.50 m and 50 increments
  - Pipe - 2 with L = 0.15 m and 15 increments
  - Fixed pressure boundary at Node-3

0.85 m total length

Acoustic Wave Propagation

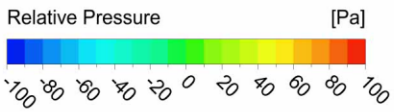






- Compressible transient simulation in Flownex and ANSYS CFX with air
- Gaussian pressure pulse specified in ANSYS CFX at inlet
- ANSYS CFX gives mass flow and average temperature to Flownex
- Flownex gives pressure and temperature to ANSYS CFX
- 1600 time steps à 5 µs → 8 ms simulation time → 2.8 m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

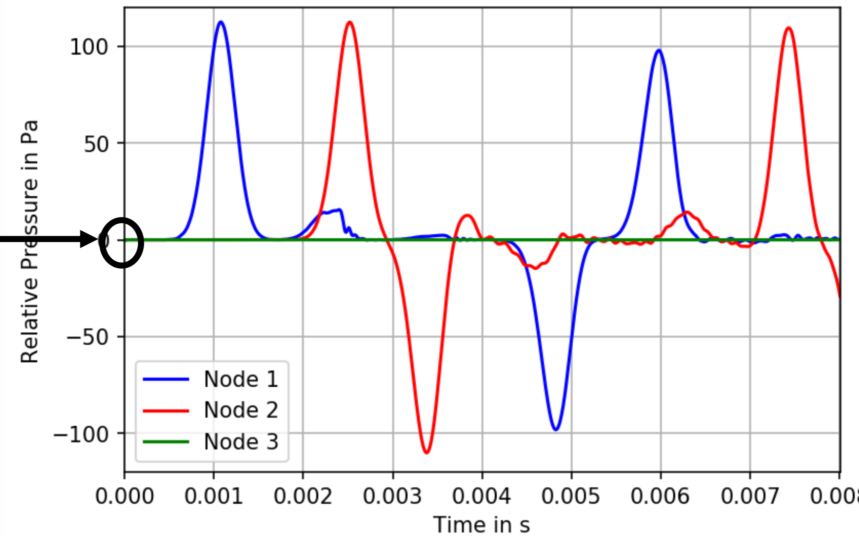
Acoustic Wave Propagation



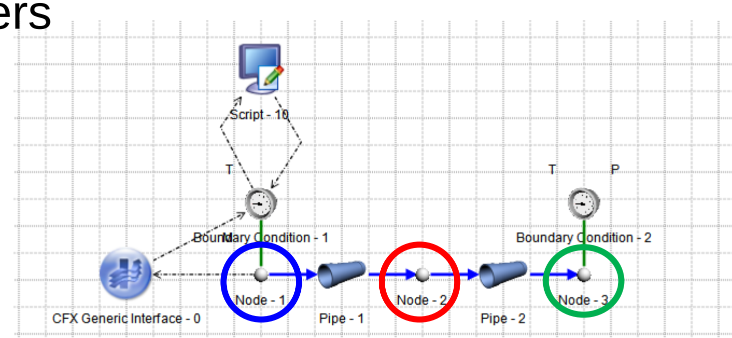
ANSYS CFX

Time = 0.00000 s

Flownex



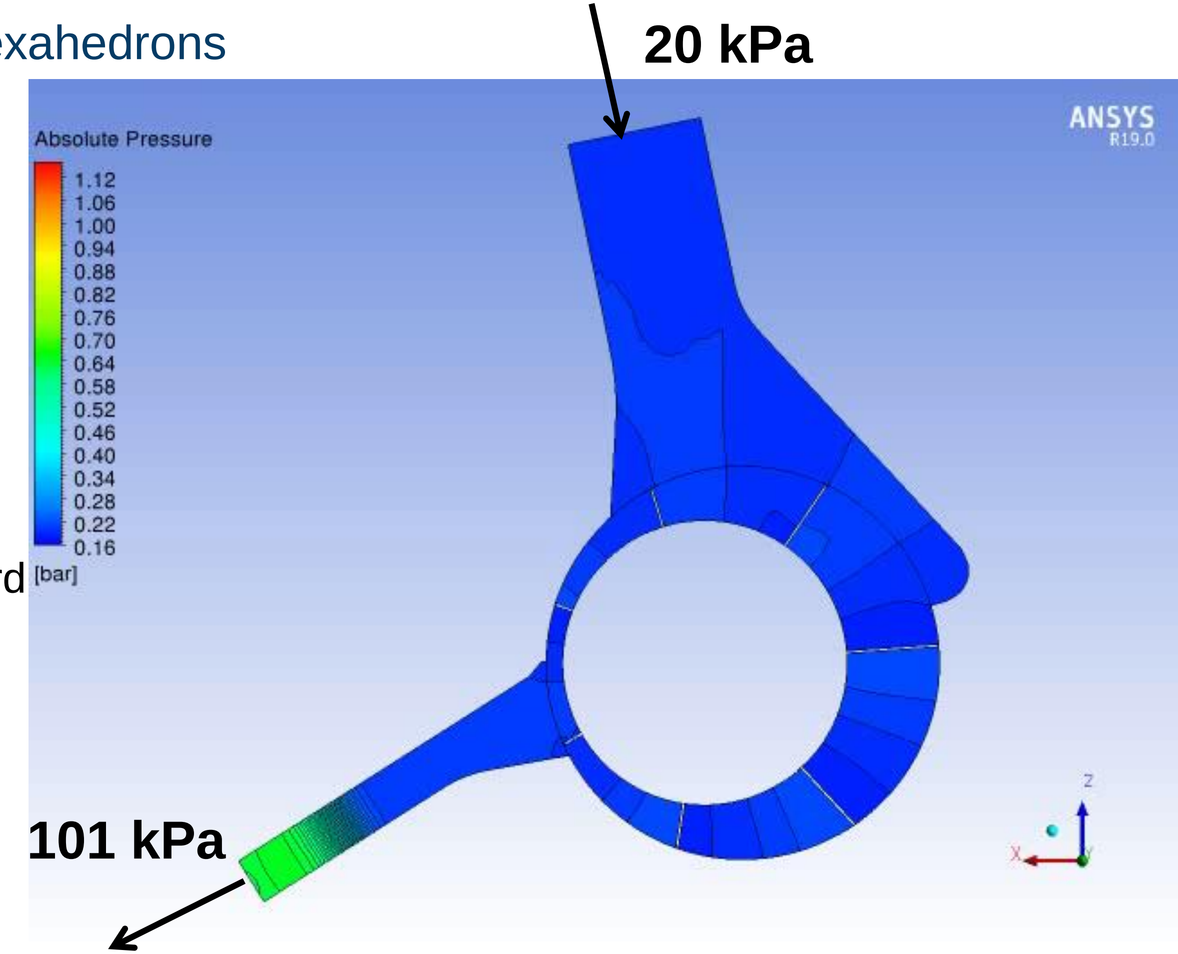
- Pressure pulse leaves ANSYS CFX domain and enters Flownex at 1 ms
- Travels towards Flownex' boundary, is reflected at Node-3 and travels back
- Enters ANSYS CFX domain at 5 ms, travels towards inlet, is reflected and travels again to right
- Enters Flownex at 6 ms…

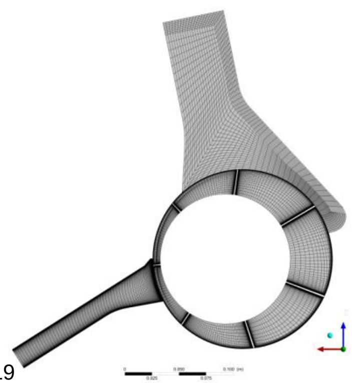


Vane Pump

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: $2380\ \mathrm{rpm}$
  - Inlet at $p_{in} = 20\ \mathrm{kPa}$
  - Outlet at $p_{out} = 101.325\ \mathrm{kPa}$
  - Openings specified as standard (reflective) boundaries

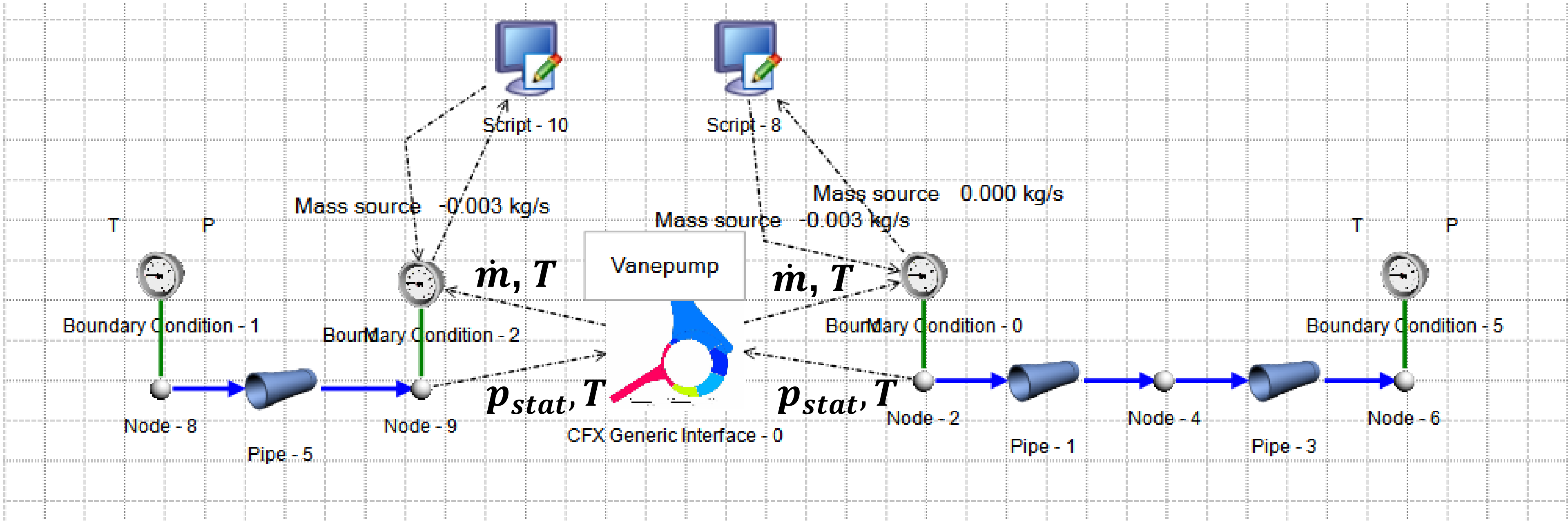




Vane Pump

CFX BERLIN

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)

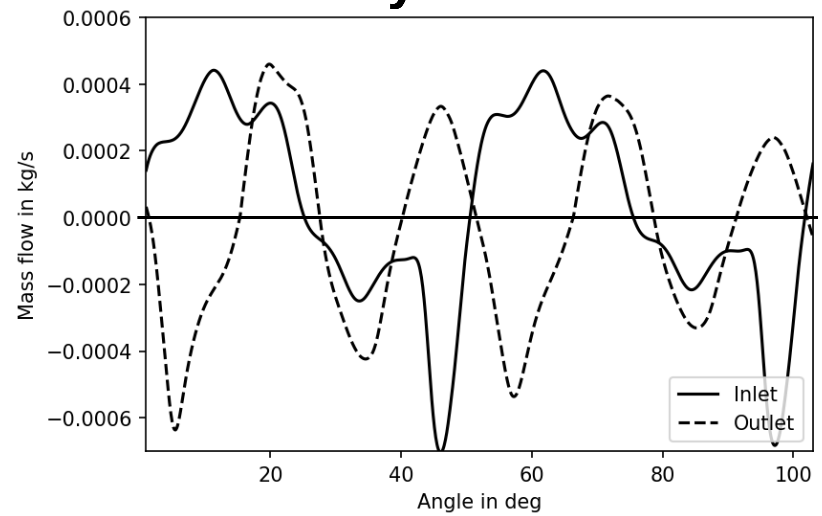
Vane Pump

CFX BERLIN

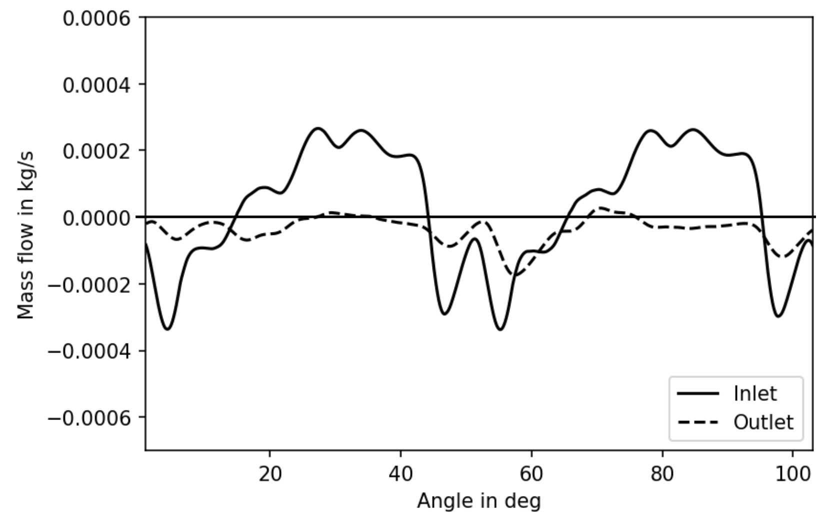
Comparison of results:

- 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

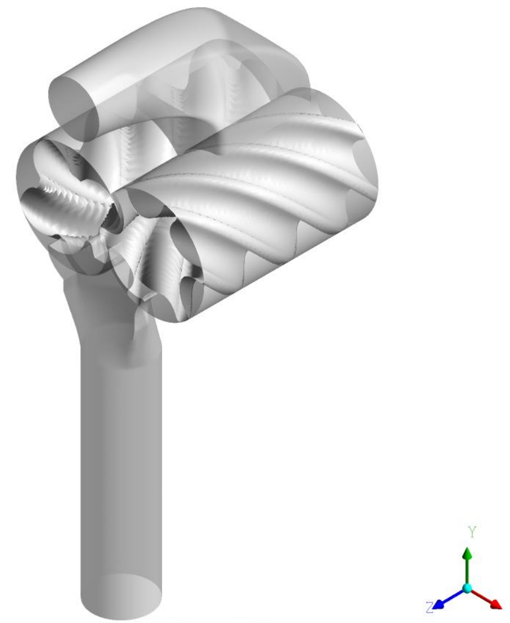


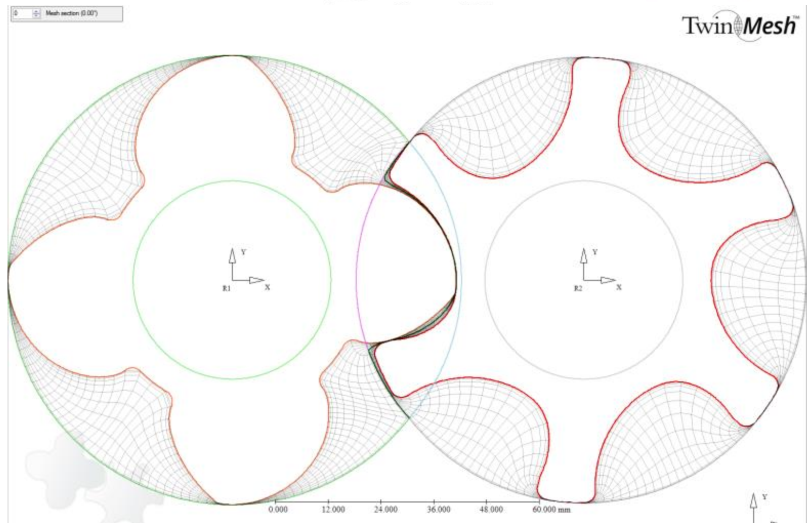
Screw Compressor

CFX BERLIN

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 $p_{in} = 1\ \mathrm{bar}, \mathrm{T} = 20°\mathrm{C}$
  - Outlet coupled to Flownex
- Time step size 68 µs (for 5° increment)
- approx. 2 h simulation time for one revolution on 4 cores

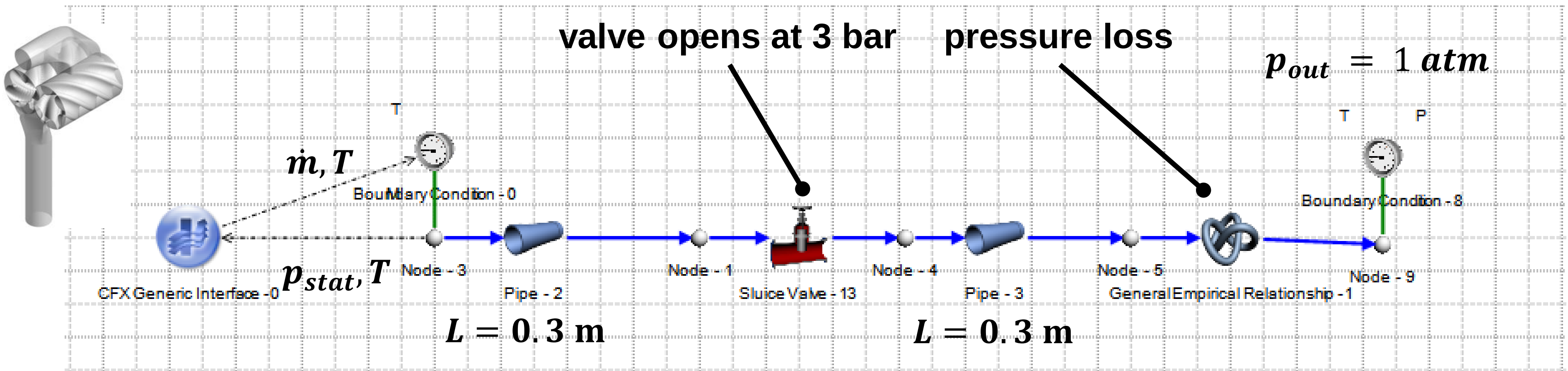




Screw Compressor

CFX BERLIN

Flownex system with ANSYS CFX co-simulation:



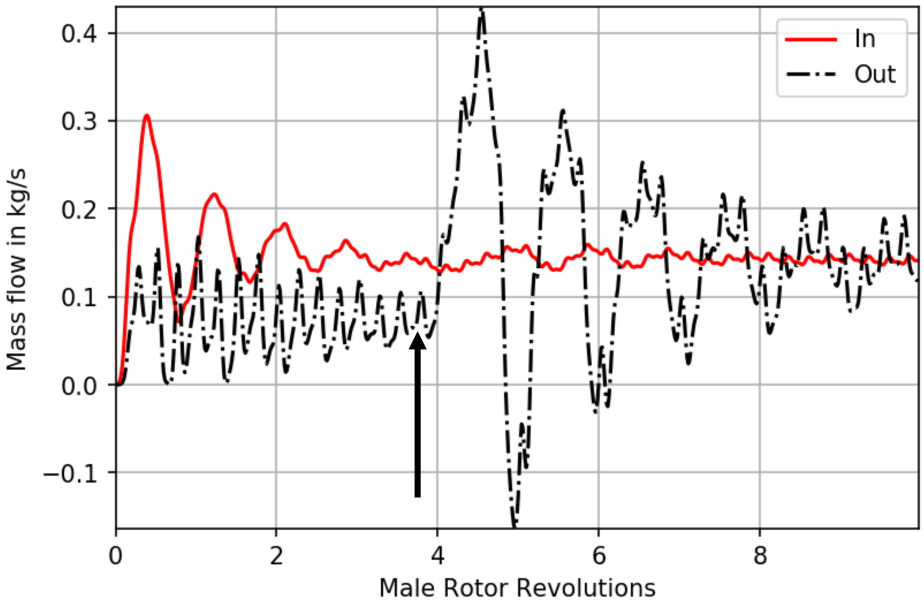
- Flownex coupled to outlet of screw compressor
- Whole system initialised at 1 bar with (almost) closed valve
- Valve opens at 3 bar (design pressure ratio 1:3 for screw compressor)

Screw Compressor

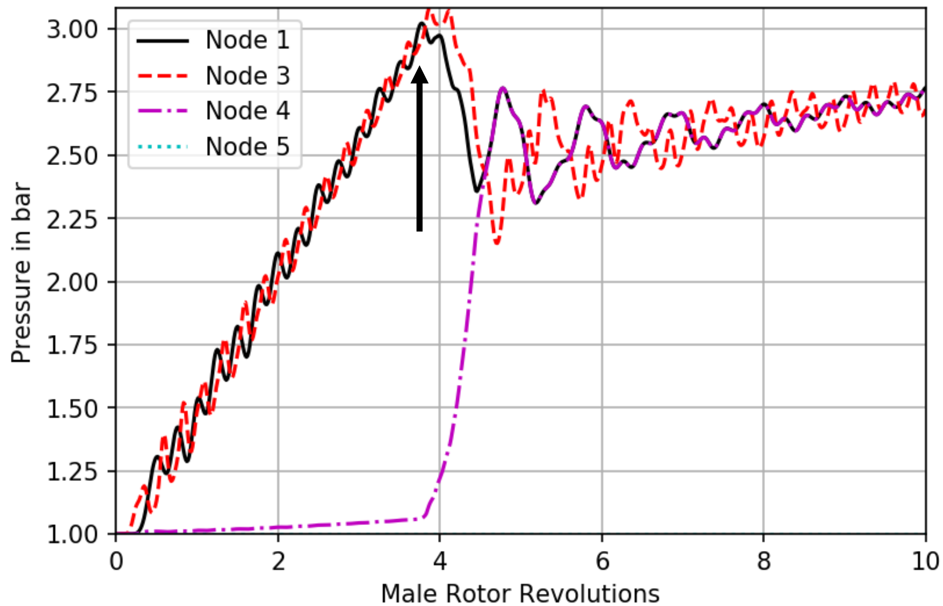
CFX BERLIN

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

Summary and Outlook

CFX BERLIN

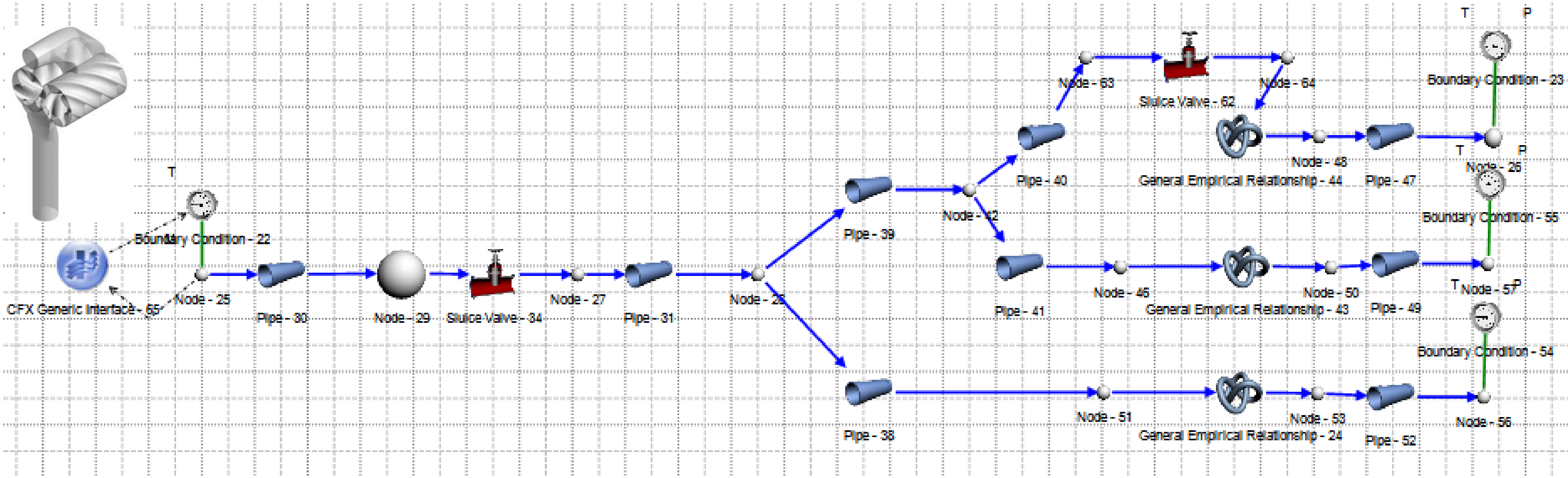
- 3D CFD simulation of positive displacement machines
  - Established tool for design and optimization
  - Fine meshes, complex physics, fast rotation, transient behaviour may require long simulation times
  - 3D CFD focuses on component with artificial boundaries
- 1D CFD allows fast simulation of attached 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

Summary and Outlook

CFX BERLIN

Example for more complex Flownex system with ANSYS CFX co-simulation:



For more information: Visit us at our stand here at the conference!