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
Contents

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

1D network model

Component
3D CFD

- 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

Conservation of mass, momentum, and energy

- 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
Flownex has a 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
• **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
• 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
• Pressure pulse leaves ANSYS CFX domain and enters Flownex at 1 ms
• Travels towards Flownex’s 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 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 $p_{in} = 20$ kPa
  - Outlet at $p_{out} = 101.325$ kPa
  - Openings specified as standard (reflective) boundaries
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)
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
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, $T = 20^\circ$C
  - Outlet coupled to Flownex
- Time step size 68 µs (for 5° increment)
- approx. 2 h simulation time for one revolution on 4 cores
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)
Results for Flownex system with ANSYS CFX co-simulation:

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

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