ANSYS CFD COUPLING TO 1D CODES AMESIM & GT-POWER AND EXAMPLES OF 1D-3D COUPLING

Dominik Scholz, Thomas Frank, Samir Patel

ANSYS Germany, Staudenfeldweg 12, 83624 Otterfing, Germany, dominik.scholz@ansys.com

SUMMARY

For the ANSYS CFD codes CFX and Fluent coupling links to several 1D codes exist allowing for a wide range of multi-physics applications. As an example, a flexible coupling infrastructure in terms of supported multiphysics coupling conditions between ANSYS CFX and LMS AMESim is outlined. Examples of the wide range of coupling solutions of ANSYS CFD (CFX and Fluent) and coupled 1D-3D applications will be presented at the seminar.

1: Co-Simulation and Flexibility in Coupling Conditions

System codes or 1D codes can compute the transient behavior of complete physical systems, e.g. hydraulic, mechanical, thermal or electrical systems. Complex systems, e.g. hydraulic pipe networks, are composed of components like pipes, valves, pumps etc. Each component is described by analytical 1D models or tabulated models. CFD (Computational Fluid Dynamics) on the other hand describe a physical system by resolving the physical phenomena in three dimensions based on solved 3D conservation & transport equations. A coupling between both approaches can be strongly desired since system codes are very limited in describing complex, especially three-dimensional, behavior of components, but a full 3D CFD simulation of a complete system might be far too demanding in terms of computational resources.

For the ANSYS CFD codes CFX and Fluent coupling links to several 1D codes exist allowing for a wide range of multi-physical applications. In the following, the coupling of the 3D CFD code ANSYS CFX [1] and the system code LMS AMESim [2] is considered as described in [3].

In order to support a wide range of multi-physics, a certain infrastructure in the participating codes is necessary:

ANSYS CFD COUPLING TO 1D CODES AMESIM & GT-POWER AND EXAMPLES OF 1D-3D COUPLING

Within the1D code it must be possible to provide any scalar results to an external code (CFD) and to set any results of the external code as an appropriate boundary condition in the 1D model. In AMESim a very flexible interface exists, where dedicated interface blocks can send any data which can be measured at sensor ports of the respective model to an external code. Arbitrary scalar data from an external code can be received at these interface blocks which in turn can be connected to any ports of the model to consider them as boundary conditions. In this way, AMESim is flexibly supporting any multiphysical coupling conditions (hydraulic, thermal, multi-phase, mechanical, etc.).

For CFD codes the requirements to the infrastructure are somewhat more complex. In order to send scalar data to the 1D code, the complex 3D data (or 2D data at interfaces) need to be reduced by averaging or integration over parts of the 3D model, e.g. the integral massflow at an inlet boundary patch or the average pressure at an outlet boundary patch need to be computed. In the other direction, scalar data from the 1D code has to be set as a boundary condition, either constant over the respective boundary patch or augmented by 3D information. As an example an integral inflow massflow can be augmented by a fully developed turbulent velocity profile shape. This infrastructure of the CFD code should support any coupling conditions in a generic way (hydraulic, thermal, mechanical, multi-component, multiphase, etc.)   to be compliant to the 1D code's capabilities.

Thus, in many CFD-1D coupling solutions, only a small subset of all possible coupling conditions is supported, e.g. only a subset of hydraulic coupling conditions is implemented (i.e. massflow provided by system code and pressure passed back by CFD), but mechanical or multi-phase coupling conditions are not supported. However, a flexible and generic infrastructure as described above was implemented in CFX, where use is made of the CFX Expression Language (CEL) [1], which can be considered as a generic run-time data access layer. In order to reduce 3D CFD data to scalars to be provided to the 1D code, CEL-expressions can be used, e.g.:

- areaAve(Pressure)@surface1

- massflow()@surface2

- force_x()@wall

- volumeAve(TracerConcentration)@subvolume1

In the other direction of data exchange, any entries in the list of scalar 1D code results can be accessed in the CFD model with CEL-

expressions which in turn can be set as boundary values for any arbitrary boundary condition type. For example, if the expression AveVelocityPatch1 contains the corresponding 1D result, one could write for boundary Patch1 being of inlet boundary type:

Normal Speed = AveVelocityPatch1 [m s^-1]

The scalar 1D results can also be augmented by 3D information. E.g., if the CEL-expression normProfile describes a fully turbulent flow profile, normalized such that its average value is unity, one can write:

Normal Speed = AveVelocityPatch1 * normProfile [m s^-1]

2: Co-Simulation Example: Tracer Transport in a Pipe Network

In order to demonstrate the efficiency of coupling 1D and 3D, i.e. using a 1D code for well-understood components and CFD where complex 3D behavior occurs, the following test case is simulated and compared to experimental data, see [4] for details on the experimental setup. The pipe network consists of an inlet, an outlet and a side loop as shown in Figure 1. The volume flow rates of water through inlet, outlet and in the side loop are controlled by pumps to be each 80l/min. At t=0s a tracer is injected into the system, travelling into the TT-junction and splitting into one part flowing through the outlet out of the system and one part re-entering the side-loop. The split of the tracer concentration is dependent on the 3D turbulent flow in the T-junction, which can hardly be predicted by 1D code. Figure 2 shows experimental measurements of the tracer concentration at wire mesh sensor WM2.
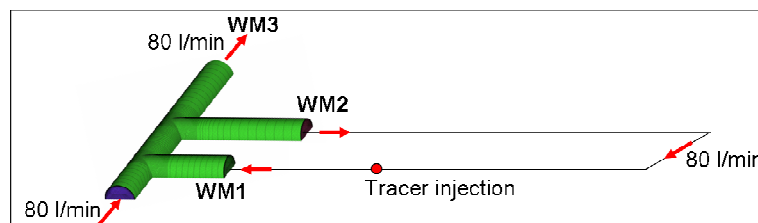

Figure 1: Pipe network with TT-Junction

An AMESim model was calibrated to model the diffusive transport of the tracer through the side loop. Two simulation runs were performed: A standalone AMESim simulation assumed the split of the tracer between system outlet and side loop to be based on the mass flow ratio, i.e. 1:1. As being obvious from Figure 2 this results in a strong underestimation of the amount of tracer re-entering the side loop. In order to properly capture the 3D flow effects, a coupled CFX-AMESim simulation was performed, where the TT-Junction was modeled with CFD and the

integral tracer concentration at WM1 and WM2 are exchanged between CFX and AMESim. The coupled simulation results show excellent agreement with the experiment.
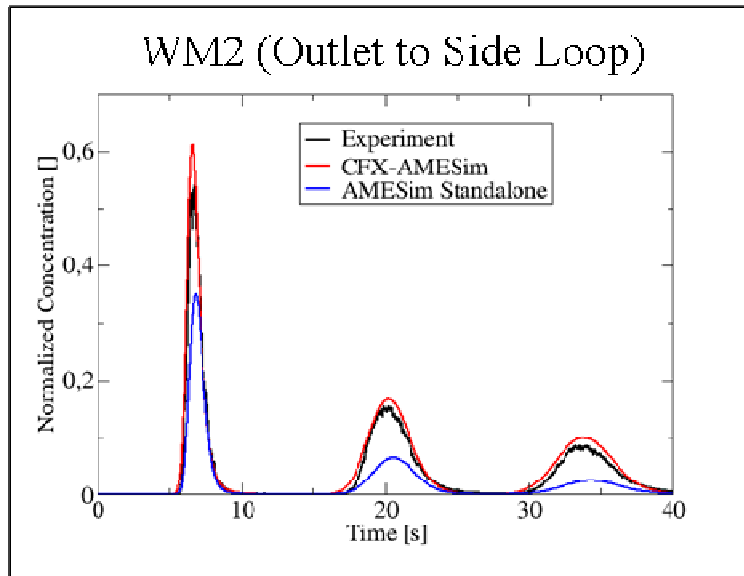

Figure 2: Tracer concentration measured at wire mesh sensor WM2

Furthermore the seminar presentation will cover examples of the wide range of coupling solutions of ANSYS CFD (CFX and Fluent) and multi-physics applications, where main areas of application are in automotive, oil & gas, chemical & process technology and power generation industry.

REFERENCES

[1]   ANSYS Inc, ANSYS CFX 12.0 Modeling Guide, 2009.

[2]   LMS International, AMESim 8A UserGuide, 2009.

[3]   D. Scholz, M. Kuntz, T. Frank, Generic Coupling Interface between ANSYS CFX and the 1D System Code LMS AMESim, ANSYS Conference & CADFEM Users' Meeting, Leipzig, 2009.

[4]   D. Bertolotto. et al., Single-phase mixing studies by means of a directly coupled CFD/system-code tool, Ann. Nucl. Energy, doi: 10.1016/j.anucene.2008.11.027, 2009.