

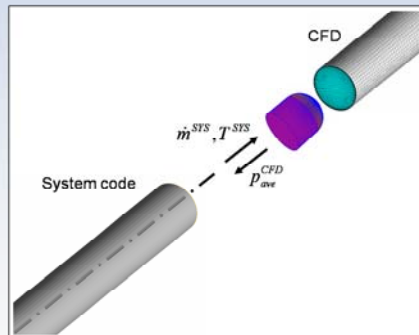
## Summary

A generic CFX-1D coupling interface is presented which allows to export scalar data to a 1D system code and to import system code results as CFX model parameters. The applicability of the generic interface is demonstrated by coupling CFX to the system code LMS AMESim.

From a user point of view, any single-value CEL expressions can be exported from CFX to the system code. Scalar system code results can be applied anywhere, where CEL expressions are allowed in a CFX model. The import and export of scalar data in the AMESim model is facilitated using the user cosim interface allowing for import and export of any AMESim component ports. For an automated setup of the CFX part of the coupled system a GUI extension is presented.

Three testcases show a broad field of application for the generic CFX-1D interface.

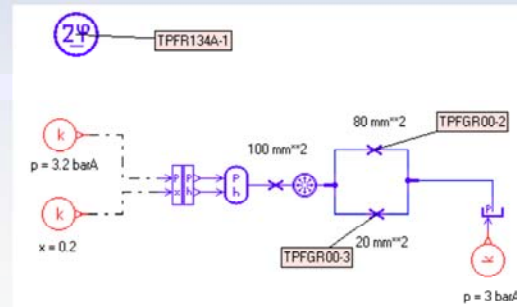
- System codes and CFD-1D coupling
- CFX-AMESim Coupling: Generic Scalar Exchange
- Test cases
  - Ball valve
  - Pipe flow
  - Tracer transport



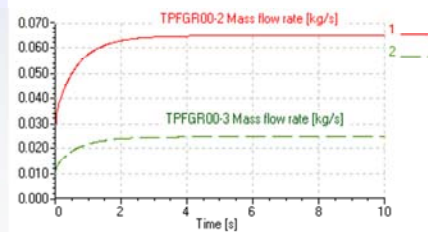
## System Codes



- Usually reduced dimensional / reduced order codes
  - transient system behaviour
  - hydraulic, mechanical, thermal, electrical, etc.
  - 1D or even 0D



AMESim example



© 2009 ANSYS, Inc. All rights reserved.

3

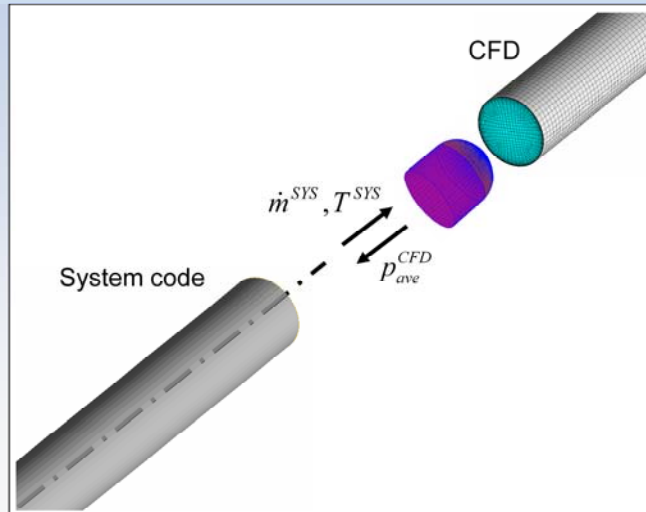
ANSYS, Inc. Proprietary

System codes are describing physical systems using reduced dimensional or reduced order models.

Examples are mechanical systems lumped to point masses which are connected with springs, 1D models or correlations for flows in pipes, etc.

The goal is to describe the transient system behavior by accounting for the interaction of its components.

## CFD-System Code Coupling



© 2009 ANSYS, Inc. All rights reserved.

4

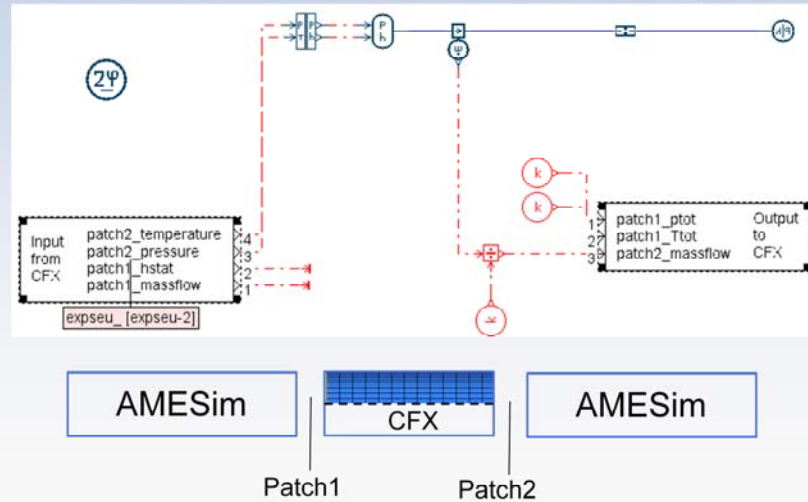
ANSYS, Inc. Proprietary

Coupling of system codes and CFD codes can be highly interesting for the simulation of complex systems. Well understood components are computed efficiently and with relatively small computational effort using system codes, whereas components with complex behavior are computed with 3D CFD models. In order to consistently couple both system code and CFD, integral or average values such as the mass flow or average pressure at interface patches are exchanged. The scalar information provided by the system code might be augmented by additional 3D information, e.g. velocity profile shapes, to be used by the CFD code.

## Coupled CFX-AMESim Model Example



- AMESim Sketch with UserCosim components



© 2009 ANSYS, Inc. All rights reserved.

5

ANSYS, Inc. Proprietary

This example of a coupled CFX-AMESim system with two coupling patches shows some of the requirements for a flexible coupling infrastructure. The simulation quantities in the AMESim model are connected to so-called UserCosim blocks with numbered ports. These have to receive an ordered set of scalar values from CFX ("Input from CFX") and have to send an ordered set of scalars to CFX ("Output to CFX").

## CFX-AMESim Coupling: Desired Functionality



- **Multiphysics application require broad range of coupling conditions (1D↔3D)**
  - $p_{\text{stat}}$ ,  $p_{\text{abs}}$ ,  $p_{\text{tot}}$ , integral forces, ...
  - $u_{\text{ave}}$ , mass flow, volume flow, ...
  - $T$ , heat flux, ...
- **At beginning of each timestep**
  - CFX calls AMESim
    - with an ordered set of scalars as arguments (e.g. integral massflow at inlet, etc.)
  - AMESim returns an ordered set of scalars
    - to be set as model parameters in CFX (e.g. pressure boundary condition, etc.)

- **CFX Expression Language (CEL)**

- Generic interface to access data in CFX

- Examples

- `massFlow() @ inlet,`
- `areaAve( Absolute Pressure ) @ outlet,`
- `massFlow() @ inlet / areaAve( Density ) @ inlet * 1e3`
- `massFlowAve( H2.pH2.Mass Fraction ) @ outlet`


fluid

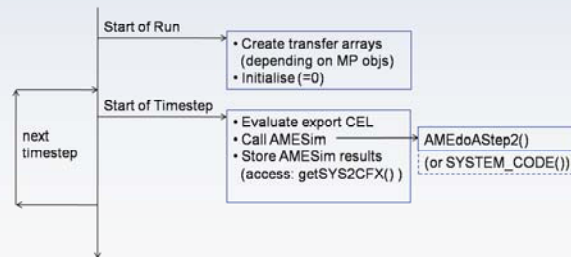
component

In order to be flexible in the choice of coupling quantities to be sent from CFX to the system code, the CFX Expression Language (CEL) can be used. As shown in the examples, all different kinds of scalars representing integral or average physical quantities over coupling patches can be evaluated. Fluid and component information can be accessed for multi-phase and multi-component calculations. Unit conversions can be accounted for by multiplication with appropriate conversion factors.

## Generic Scalar Exchange: Approach



- Synchronized call of AMESim time advancing routine
  - Send an ordered set of arbitrary single-value CEL-expressions
  - At each start of timestep (once per timestep, explicit)
- Set results of system code where CEL-expressions are allowed: 



© 2009 ANSYS, Inc. All rights reserved.

8

ANSYS, Inc. Proprietary

The basic approach is to call the system code's time advancing routine with an ordered set of arbitrary single-value CEL-expressions as arguments. This is to be done once at the start of each CFX timestep resulting in an explicit scheme. In the opposite direction, it should be possible to set the results of the system code everywhere, where CEL-expressions are allowed (depicted symbol) which works for almost all boundary conditions, sources, system parameters, etc. Thus, a very generic coupling infrastructure is provided.



## Generic Scalar Exchange: Ingredients



- **Monitor point objects embedding CEL-expressions for each export/import variable:**

- `MP_CFX2SYS_1[_opt_descr],`  
`MP_CFX2SYS_2[_opt_descr], ...`

- `MP_SYS2CFX_1[_opt_descr],`  
`MP_SYS2CFX_2[_opt_descr], ...`

- **User Fortran coupling library for Linux64**

- Junction Boxes
    - Start of Run, Start of Timestep
  - User-CEL `getSYS2CFX(int)`
    - To access scalars from system code in CFX
  - Support of parallelization

- **GUI-Extension for automated setup**

The approach discussed in the previous slide was implemented as a User Fortran coupling library for Linux64. Monitor point objects with a specific naming convention are used to embed the CEL-expressions to be sent from CFX to the system code. Also the system code results are monitored accordingly. In order to set the system code results within the CFX model, the User Fortran CEL-function `getSYS2CFX(int)` was implemented. Parallelism is accounted for. GUI-Extensions were implemented to support the user in the setup of monitor point objects, User Fortran routines, etc.

## Example 1: Ball Valve



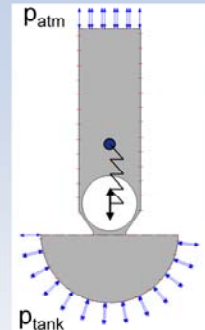
- **Tutorial example**
  - FSI
  - Motion of rigid ball as spring-mass system (2nd order ODE in time)

- **Comparison: different time integration schemes**

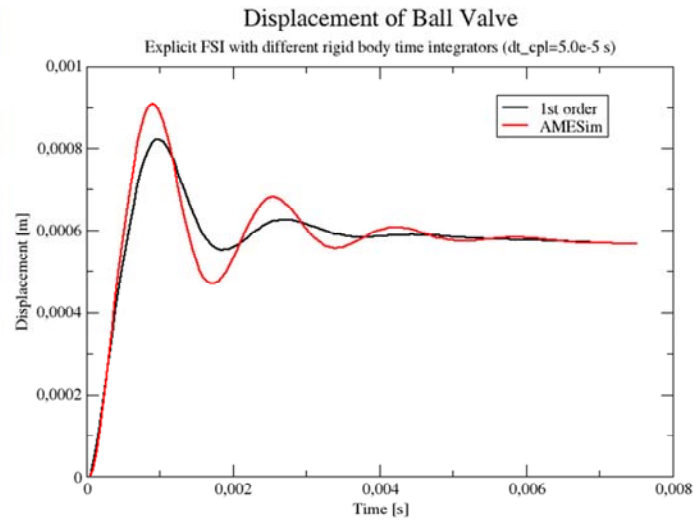
- 1st order (as in tutorial, CEL-based)

$$\left(\frac{dv}{dt}\right)_{ball}^{new} = \frac{v_{ball}^{new} - v_{ball}^{old}}{\Delta t} \quad v_{ball}^{new} = \frac{d_{ball}^{new} - d_{ball}^{old}}{\Delta t}$$

- AMESim (coupled via GSX)
    - 2nd or higher order schemes (e.g. Newmark, Runge-Kutta, etc.)



## Example 1: Ball Valve



© 2009 ANSYS, Inc. All rights reserved.

11

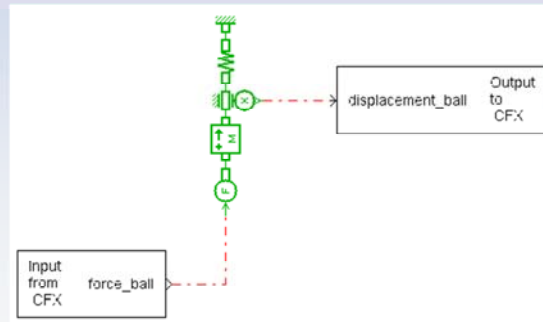
ANSYS, Inc. Proprietary

The results show that the 1st order time discretization of the rigid body motion of the ball is clearly more dissipative than the algorithm used by AMESim. Both simulations obviously reach the same stationary state. In the following the setup of the coupled system is discussed.

## Example 1: Ball Valve

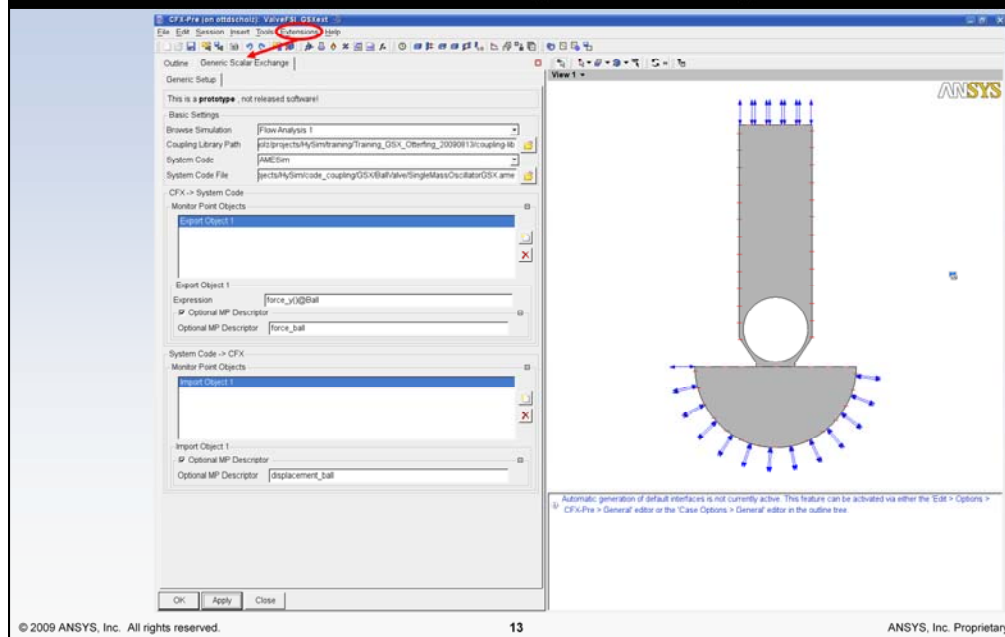


- AMESim model



The AMESim model consists of the mass, of the spring fixed on one side, and of the UserCosim blocks receiving the force from CFX and sending the displacement to CFX.

## Example 1: Ball Valve GUI Extension for Coupled Setup

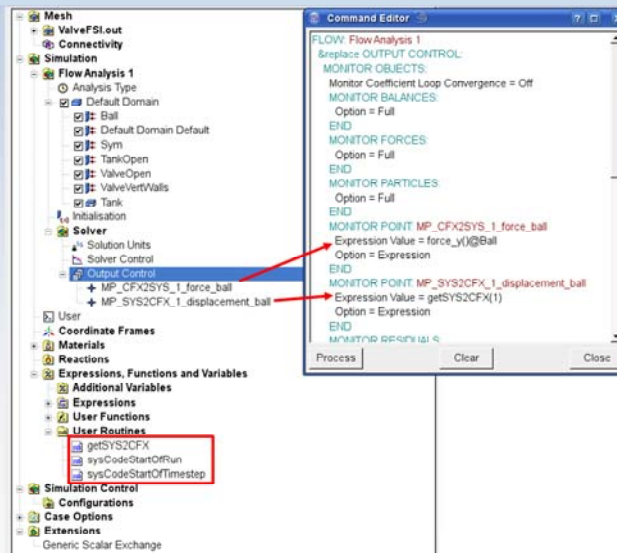


A so-called GUI-Extension was implemented to support the user in the setup phase of the coupled simulation. General data such as the Configuration (Simulation), the path to the coupling library, the system code and the system code model file can be set. For the definition of the coupling conditions, monitor point objects for export (with CEL expressions) and for monitoring the imported data have to be specified. Optional descriptors can be provided to get readable monitor point names. By applying the data in the dialog, the CFX model is extended by the coupling setup, see the following slides.

## Example 1: Ball Valve CFX Model Tree

ANSYS

- CFX Model Tree



© 2009 ANSYS, Inc. All rights reserved.

14

ANSYS, Inc. Proprietary

The model tree (left) shows the monitor point objects which have been created for the data exchange after applying the GUI extension dialogue. Furthermore, the User Fortran Junction Box Routines and CEL-Functions are integrated into the model.

## Example 1: Ball Valve



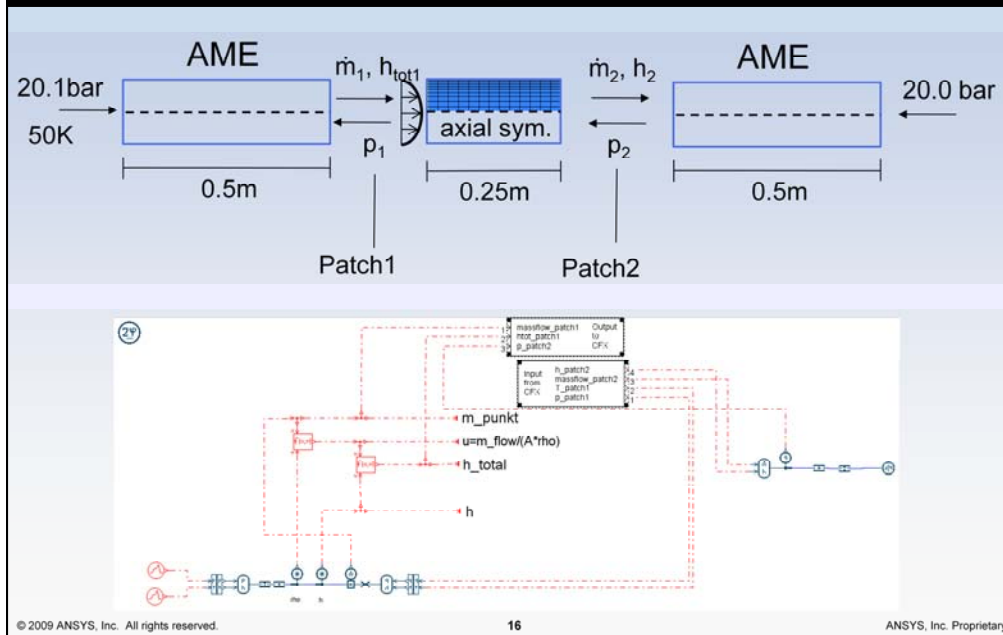
- Boundary conditions provided by system code

The screenshot shows the 'Boundary: Ball' panel in ANSYS Fluent. The 'Basic Settings' tab is active. Under 'Mass And Momentum', the 'Option' is 'No Slip Wall', 'Wall Velocity Relative To' is 'Mesh Motion', and 'Wall Velocity' is set to 0. Under 'Wall Roughness', the 'Option' is 'Smooth Wall'. Under 'Mesh Motion', the 'Option' is 'Specified Displacement'. The 'Displacement' section shows 'Option' as 'Cartesian Components'. The 'X Component' is 0 [m], the 'Y Component' is `getSYS2CFX(1) [m]` (highlighted with a red box), and the 'Z Component' is 0 [m].

The last thing the user has to do is to set the system code results where desired. In this example (right) the one and only system code result, the displacement of the ball, is set as a mesh displacement of the corresponding boundary patch using `getSYS2CFX(1)`.

## Example 2: Adiabatic Pipe $\varnothing=5\text{mm}$ Cryogenic $\text{H}_2$

ANSYS



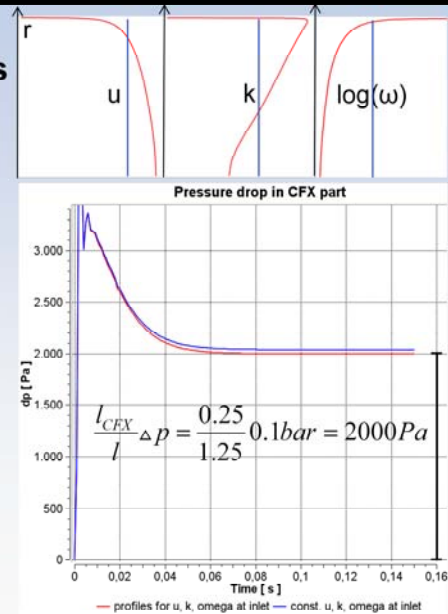
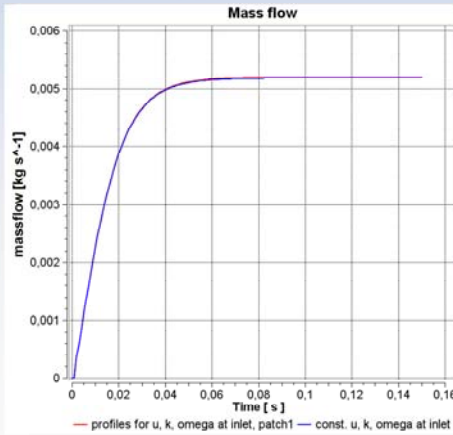
The second example is validation test case for pipe flow. The pipe is divided into two AMESim parts and one CFX-part, where the length of the CFX-part is a fifth of the total length. The testcase should indicate whether the pressure gradients computed by AMESim using the Churchill correlation and those computed by CFX using the SST turbulence model are the same, in order to ensure consistency of the physical models in use.



# Mass Flow and Pressure Drop Profiles Close "1D-3D Gap"



- **Const. vs. developed profiles**
  - appropriately scaled  $u$ ,  $k$ ,  $\omega$  at CFX inlet



© 2009 ANSYS, Inc. All rights reserved.

17

ANSYS, Inc. Proprietary

In the CFX-model, the viscous sub-layer was resolved for all simulations ( $y^+ = 2$ ) and the SST turbulence model was used.

Using block profiles (blue) for the velocity, the turbulent kinetic energy and the turbulent frequency the pressure drop is slightly overestimated in comparison to Churchill's correlation for the pressure drop used by AMESim.

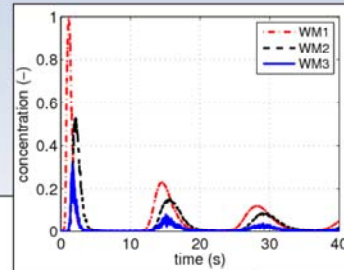
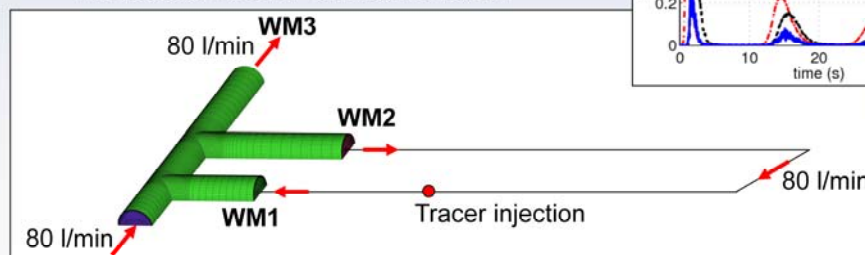
In contrast, using appropriately scaled developed profiles (red) leads to the expected pressure drop according to the ratio of the length of the CFX part to the total length. In other words, the CFX model and the AMESim model compute the same pressure gradients.

## Example 3: Tracer Transport



- **Testcase Tracer Transport (Bertolotto et al., 2009)**

- Inlet, outlet, side loop
- 1:1 mass flow ratio outlet/side loop
- Small amount of tracer injected
- Wire mesh sensors WM1-WM3 measure tracer concentration



© 2009 ANSYS, Inc. All rights reserved.

18

ANSYS, Inc. Proprietary

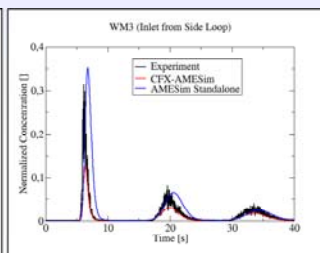
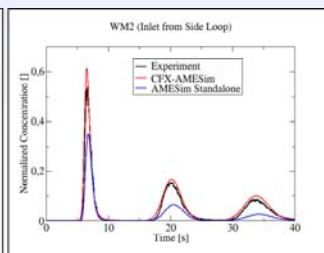
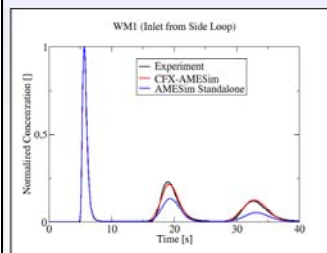
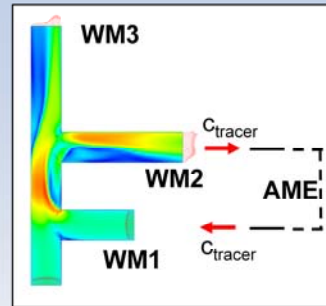
The following test case is described in the article: Bertolotto, D. et al., Single-phase mixing studies by means of a directly coupled CFD/system-code tool, Ann. Nucl. Energy (2009), doi: 10.1016/j.anucene.2008.11.027.

The system is shown in the figure on the bottom. On the right hand side the experimental values for the transient tracer concentrations measured at the wire mesh sensors are plotted (normalized to a maximum value of 1).

## Example 3: Tracer Transport



- **Coupled CFX-AMESim model**
  - 3D effects in TT-junction represented
- **Standalone 1D AMESim model**
  - Tracer split only according to mass flow ratios (1:1)



© 2009 ANSYS, Inc. All rights reserved.

19

ANSYS, Inc. Proprietary

An AMESim model was calibrated to model the diffusive transport of the tracer through the side loop (calibration with first peak at WM2 as input, transported to second peak at WM3). 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 the plots this results in a strong underestimation of the amount of tracer entering the side loops.

In order to properly capture the 3D flow effects, a coupled CFX-AMESim simulation was performed, where the TT-Junction was modeled with CFD. The coupled simulation results show excellent agreement with the experiment.

- **CFX-AMESim Coupling: Generic Scalar Exchange**
  - User Fortran based coupling library on Linux64
- **GUI-Extensions to support user in setup phase**
- **Test cases show broad range of multiphysics applications:**
  - Ball valve: fluid-structure-interaction
  - Pipe flow: 1D results augmented by scaled profiles
  - Tracer transport: comparison with experiment