PROTOTYPE COUPLING OF THE CFD CODE ANSYS CFX WITH THE 3D NEUTRON KINETIC CORE MODEL DYN3D

S. Kliem, U. Rohde
Forschungszentrum Dresden-Rossendorf, Institute of Safety Research

J. Schütze, Th. Frank
ANSYS Germany GmbH

1. Introduction

Analyses of postulated reactivity initiated accidents in nuclear reactors are carried out using 3D neutron kinetic core models. The feedback is usually calculated using 1D thermal hydraulic models for channel flow, partly with the possibility of cross flow between theses channels. A different possibility is the use of subchannel codes for the determination of the feedback. The code DYN3D developed at Forschungszentrum Dresden-Rossendorf is an example for a 3D neutron kinetic core model. In its basic version, the code contains models for the solution of the 3D neutron diffusion equation in two energy groups for fuel assemblies with rectangular and hexagonal cross section [1]. Recently the code was extended to an arbitrary number of energy groups. Further, a simplified transport approximation for the flux calculation was implemented for fuel assemblies with quadratic cross section [2].

The CFD code ANSYS CFX [3] is the reference CFD code of the German CFD Network in Nuclear Reactor Safety. One of the goals of the co-operation inside this network is the development of CFD software for the simulation of multi-dimensional flows in reactor cooling systems. This includes the coupling of the CFD code ANSYS CFX with the 3D neutron kinetic core model DYN3D.

2. Coupling of ANSYS CFX and DYN3D

The coupling approach is based on the selection of best-in-class software tools for the simulation of each of the phenomena to be described by the coupled codes. For this, the module predicting the coolant flow within DYN3D is replaced by a fully three-dimensional CFD simulation using ANSYS CFX. A detailed and spatially resolved modeling of the whole reactor core down to the fuel pin level in the CFD code is not feasible for practical applications at present and in the foreseeable future. It is possible to achieve acceptable computation times only by modeling the reactor core as a porous region. This reduced resolution of the structures in the core affects the thought location of the interface between the CFD code and the neutron kinetics core model. An incorporation of the bare neutron kinetics model of DYN3D only, as it was done in the internal coupling of ATHLET and DYN3D [4], is not possible because the heat transfer from the fuel pins to the coolant cannot be calculated by ANSYS CFX due to the above mentioned restrictions. Therefore, it was decided to define the physical data interface at the level of the volumetric heat release rate into the fluid. The CFD code ANSYS CFX calculates the fluid dynamics in the reactor coolant inside the core. It provides the velocity, temperature, density and boron concentration

fields to DYN3D. Based on these parameters DYN3D determines the nuclear power, calculates the fuel temperature distribution and the heat transfer to coolant. The volumetric heat source is given back to ANSYS CFX. It should be noted that in the current prototype, the coupling is restricted to single-phase flow conditions.

In the coupled calculation, ANSYS CFX acts as the master program; DYN3D is implemented as a set of subroutines. A 3D volume mesh-to-mesh transfer of field quantities between ANSYS CFX and DYN3D had to be implemented taking into account the largely different mesh resolutions used in the two codes. A 3D-volume mesh-to-mesh transfer for arbitrary data fields was implemented in CFX. The conservation of the data during transfer is properly ensured. The DYN3D coarse nodalisation is represented by a separate, coarsely meshed zone in CFX. This coarsely meshed zone is also available for post-processing DYN3D data in CFX-Post. Both zones co-exist side-by-side in CFX.

For steady-state calculations an iteration scheme between ANSYS CFX and DYN3D was implemented. In the DYN3D stand-alone case the thermal hydraulics is brought to convergence at each iteration step before going to the solution of the neutron-kinetic equations. In the coupled code calculation the approach is different: DYN3D is called at the end of each iteration step of ANSYS CFX. In this way, the number of iterations between the codes increases, but this implementation requires less total computation time as the dominant part of the computation time is spent for ANSYS CFX.

At the current stage of the implementation no iteration between ANSYS CFX and DYN3D is carried out during transient calculation. An explicit coupling approach is applied. DYN3D is called at the end of each time-step.

3. Verification

3.1. Steady-state problem

For the verification of the steady-state calculation procedure, a mini-core consisting of nine real size PWR fuel with a power of 50 MW was set-up (Fig. 1). As usual, the resolution of the DYN3D grid is one node per fuel assembly in radial direction. 14 nodes were used over the height. The CFX calculation grid contains 14.308 nodes. The fluid flow solver in ANSYS CFX was set up so as to only allow purely 1D flow in parallel channels in order to ensure the comparability with the DYN3D stand alone results.

Fig. 2 shows the convergence of the effective multiplication factor in both calculations. The number of iterations is higher in the coupled calculation. This is due to the above mentioned different iteration scheme, where the neutron-kinetic module is called at every iteration step in CFX. In the stand alone version of DYN3D, the neutron-kinetic module is called only when the thermal hydraulics has reached convergence. Here, the internal thermal hydraulic iterations are not counted. The resulting Keff-values differ by 9.8 pcm, only. The reason for the differences was found in different material property packages. CFX uses the current standard for water properties IAPWS-IF97 while DYN3D uses the former standard IFC-67.

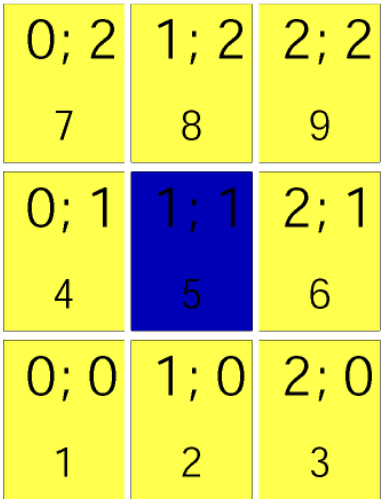

Fig. 1: Scheme of the mini-core

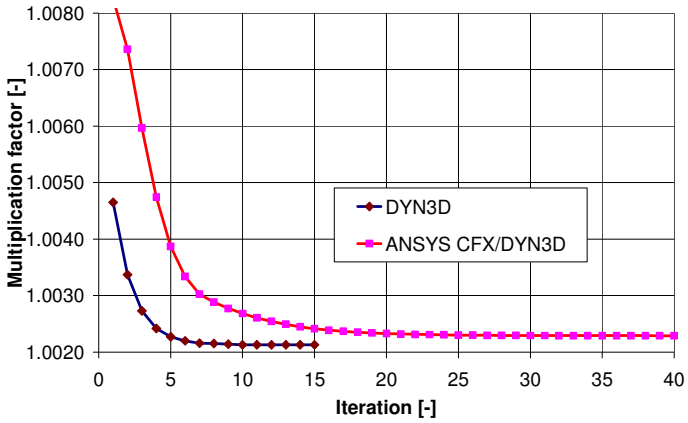

Fig. 2: Convergence of steady-state calculations

3.2. Transient problem

For the verification of the implementation of the transient calculation option the withdrawal of the control rod from the central fuel assembly at hot zero power was selected. The initial position of the control rod was selected with 1.50 m from lower edge. The time for full withdrawal was set to 20 s. Time-explicit coupling had been implemented so far, only. Calculations were carried out with variation of the time step size. The results of the calculations with time steps of 10 and 1 ms are shown on Fig. 3 and Fig. 4. For comparison purposes the iteration within each time step in the DYN3D stand-alone code was deactivated.

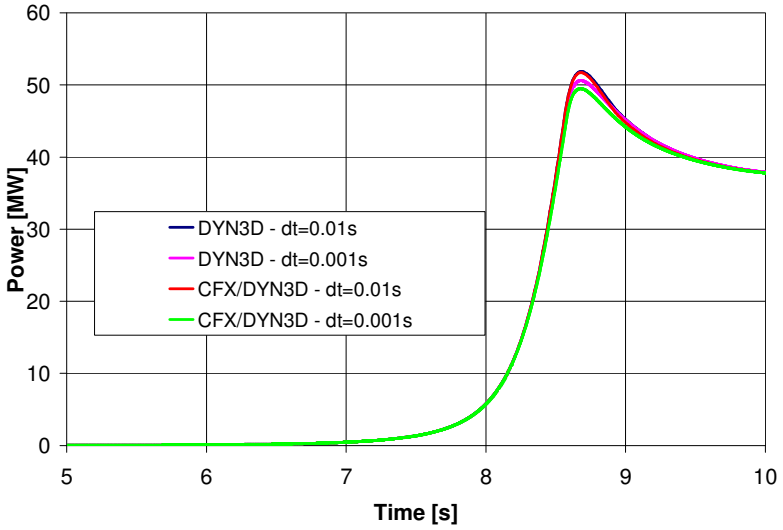

Fig. 3: Core power behavior in the DYN3D and CFX/DYN3D calculations

The power starts to rise remarkably after 7 s. A power peak occurs which is compensated by the Doppler feedback. Due to continuing control rod extraction the power will rise further. Here, only the first 10 s are analyzed. This includes the first power maximum and the power decrease due to fuel temperature increase.

The overall behavior of the core power is identical in all four calculations. Differences are to be seen in the maximum value reached. This maximum reduces with decreasing time step size in both codes. Both codes

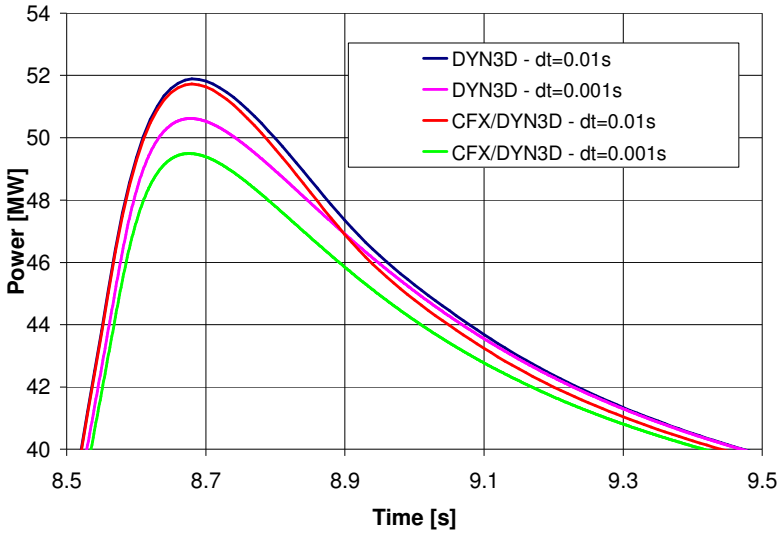

Fig. 4: Core power behavior in the DYN3D and CFX/DYN3D calculations (zoom)

converge to different maximum values. This was proven by further reduction of the time step. The difference in the maximum core power is about 1 MW. This difference is acceptable considering the following: The introduced positive reactivity during the transient is slightly below 1 $. In the reactivity range around 1 $ the power behavior is very sensitive to changes in the feedback parameters. Therefore, smallest differences between the different transient flow solvers can have significant effect on the time-dependent feedback. One such difference is already known to lie in different water material property packages.

Further investigation on the transport of a temperature perturbation through the reactor core confirmed that the DYN3D flow solver shows some additional difference in comparison to the ANSYS CFX flow solver.

4. Conclusions and future work

The coupling of the CFD code ANSYS CFX with the neutron-kinetic core model DYN3D was successfully accomplished. The new coupled code system ANSYS CFX/DYN3D allows for more realistic analyses of coupled thermal hydraulics – neutron kinetics problems. Steady-state and transient verification calculations for a small-size test problem confirmed the correctness of the implementation of the coupling.

Further verification and validation is needed before its application to accident scenarios. In the near future the semi-implicit time domain coupling should be introduced in the coupling. The extension of the coupling to two-phase flow conditions is a further precondition to carry out realistic accident analyses. In this field additional methodical work has still to be done, e.g. on the splitting of the volumetric heat source between liquid heat-up and vaporization and on two-phase flow treatment in the porous body approach. Further work will also be done on the automated mesh generation for the coupled code.

Acknowledgment

The projects this paper is based on were funded by German Federal Ministry of Economics and Technology (Contract numbers 150 1328 and 150 1358).

References

[1] U. Grundmann, U. Rohde, S. Mittag, “DYN3D – Three Dimensional Core Model for Steady-State and Transient Analysis of Thermal Reactors”, Proc. PHYSOR2000, CDROM (2000)

[2] C. Beckert, U. Grundmann, “Development and verification of a nodal approach for solving the multigroup SP3 equations”, Annals of Nuclear Energy, 35/1, pp. 75-86 (2008)

[3] ANSYS Inc., ANSYS 12.0 User’s Guide, Canonsburg, PA, USA (2009)

[4] Y. Kozmenkov, S. Kliem, U. Grundmann, U. Rohde, F.-P. Weiss, “Calculation of the VVER-1000 coolant transient benchmark using the coupled code systems DYN3D/RELAP5 and DYN3D/ATHLET”, Nucl. Eng. Design, 237, pp. 1938-1951 (2007)