

CFD-NEUTRONIC COUPLED CALCULATION OF A PWR FUEL ASSEMBLY CONSIDERING PRESSURE DROP AND TURBULENCE PRODUCED BY SPACER GRIDS

C. Peña-Monferrer^a, F. Pellacani^d, S. Chiva^{b,*}, T. Barrachina^c, R. Miró^c, R. Macián-Juan^d

^a*Institute for Energy Engineering, Universitat Politècnica de València. Camí de Vera, s/n, 46022 València, Spain*

^b*Department of Mechanical Engineering and Construction, Universitat Jaume I. Campus del Riu Sec, 12080 Castelló de la Plana, Spain*

^c*ISIRYN, Universidad Politècnica de Valencia. Camí de Vera, s/n, 46022 Valencia, Spain*

^d*Ntech Lehrstuhl für Nukleartechnik, Technische Universität München. Boltzmannstr. 15, 85748 Garching, Germany*

Abstract

A computational code system called Coupled Solver ANSYS CFX/PARCS (CSAP) based on coupling the 3D neutron diffusion code PARCS v2.7 and the ANSYS CFX 13.0 Computational Fluid Dynamics (CFD) code has been developed as a tool for nuclear reactor systems simulations. This paper presents the coupling methodology between the CFD and the neutronic code. The methodology to simulate a 3D-neutronic problem coupled with 1D thermalhydraulics is already a mature technology, being part of the regular calculations performed to analyze different kinds of Reactivity Insertion Accidents (RIA) and asymmetric transients in Nuclear Power Plants, with state-of-the-art coupled codes like TRAC-B/NEM, RELAP5/PARCS, TRACE/PARCS, RELAP3D, RETRAN3D, etc. The transport of neutrons depends on several parameters, like fuel temperature, moderator temperature and density, boron concentration and fuel rod insertion. These data are calculated by the CFD code with high local resolution and used as input to the neutronic code to calculate a 3D nodal power distribution that will be returned and remapped to the CFD code control volumes (cells). Since two different nodalizations are used to discretize the same system, an averaging and interpolating procedure is needed to realize an effective data exchange. These procedures have been developed by means of the ANSYS CFX “User Fortran” interface; a library with several sub-routines has been developed for calculation and synchronization purposes. The data exchange is realized by means of the Parallel Virtual Machine (PVM) software package. In this contribution, steady-state and transient results of a quarter of PWR fuel assembly with cold water injection are presented and compared with obtained results from a RELAP5/PARCS v2.7 coupled calculation. A simplified model for the spacers has been included. A methodology has been introduced to take into account the pressure drop and the turbulence enhancement produced by the spacers.

Keywords: CFX, PARCS, RELAP, Coupling, TH/N, CSAP

*Corresponding author

Email addresses: cmonfer@upv.es (C. Peña-Monferrer), pellacani@ntech.mw.tum.de (F. Pellacani), schiva@emc.uji.es (S. Chiva), rmiro@iqn.upv.es (R. Miró)

1. Introduction

In order to license a nuclear power plant (NPP) a broad range of analyses can be carried out by using 1D thermal-hydraulic Best Estimate (BE) codes able to simulate the entire plant in transient and accidental conditions. By using them, it is possible to simulate a wide variety of scenarios not only involving accident conditions, such as, for instance, Loss of Coolant Accidents (LOCAs), but also transients of interest for normal operation, like the insertion or extraction of control rods. These transients can be analyzed with the available coupled thermal-hydraulic-neutronic code systems which are capable of simulating the thermal hydraulic and neutronic behaviour of a nuclear reactor with a high grade of reliability. Nevertheless, the detailed study of asymmetries in the power and mass flow distributions inside the fuel assemblies, even using the coarse 3D flow capabilities available in some of the BE codes, is somehow beyond the scope of these coupled code systems. A high degree of intra-fuel assembly flow spatial resolution can be achieved with Computational Fluid Dynamics (CFD) codes, they are able to reproduce detailed 3D flow phenomena at the level of single fuel rods, and can also consider turbulence and its effect on the dynamics of the flow that determine local heat transfer phenomena of importance in the evaluation of fuel integrity. CFD codes yield very detailed velocity and temperature fields in the moderator, which can be then coupled to refined neutronic and fuel material descriptions in order to obtain an unprecedented degree of fidelity in the analysis of nuclear fuel behaviour.

Nuclear fuel bundles contain spacer essentially for mechanical stability and to influence the flow dynamic and heat transfer phenomena along the fuel rods. The CFD simulation of the turbulent phenomena downstream of the spacer grid presents high complexity issues because a wide range of length scales are present increasing the difficulty of defining in detail the transient nature of turbulent flow. Hence the computational resources are still too high to provide reasonable CFD simulations of fuel assemblies including spacer grids since the concerned nuclear reactor core contains a large number of spacer grids. A proposed methodology to take into account the main effects of spacer grids on channel flow as pressure drop and turbulence production is performed.

2. Description of the coupled system

This section briefly describes the problem to be solved by the coupled tools and the way in which the equations of the general CFD software ANSYS CFX 13.0 and the neutron diffusion code are solved using common variables. As result of the process, both programs working jointly can synchronize and use as input data the output data of the other code and viceversa. A state-of-the-art CFD code is able to predict with a high degree of accuracy the thermal- hydraulic behavior (steady-state and transient) of a solid-liquid system like that represented by a fuel assembly of a nuclear reactor when single-phase flow dynamics is considered. In previous work, the power shape considered in the nuclear fuel has been assumed to have a constant shape, or be determined by a given time dependent function, not influenced by the neutronic behavior of the fuel. A neutronics code needs the values of fundamental thermal-hydraulic variables such as moderator temperature and density, and temperatures of the solid structures in order to determine the neutron flux distribution and the power produced in the fuel in a dynamic manner. As mentioned previously PARCS is currently coupled to the 1D thermal-hydraulic code RELAP5 or the 3D code TRACE. The information transfer between those codes is basically similar to what is needed when a CFD code is

employed instead of the coarser thermal-hydraulic codes RELAP5 and TRACE. The correspondence between the thermal- hydraulic and neutronic meshes is relatively straightforward in the case of the coarser codes. Since the dimensions of the neutronic nodes and of the thermal-hydraulic computational volumes are similar, the values of the variables can be transferred between the codes relatively unchanged. In the case of using a CFD code for the flow and heat transfer simulation, however, neutronic and thermal hydraulic meshes have very different dimensions.

The CSAP coupling procedure and the first preliminary results are explained in detail in previous contributions [1] [2] [3]. In this work only the working principle and latter a proposed methodology to consider spacer grids are reported. Fig. 1 describes the data exchange flow.

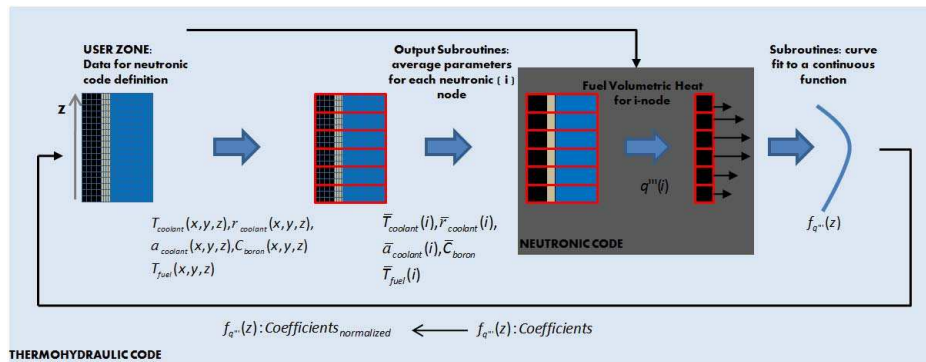


Figure 1: Data exchange flow

3. Description of the models

This section describes the problem to be solved by the coupled tools and the way in which the geometrical model has been implemented in the thermalhydraulics and neutronic codes. A quarter of the nuclear reactor fuel assembly will be examined; a simplified sketch of the model cross section is shown in Fig. 2.

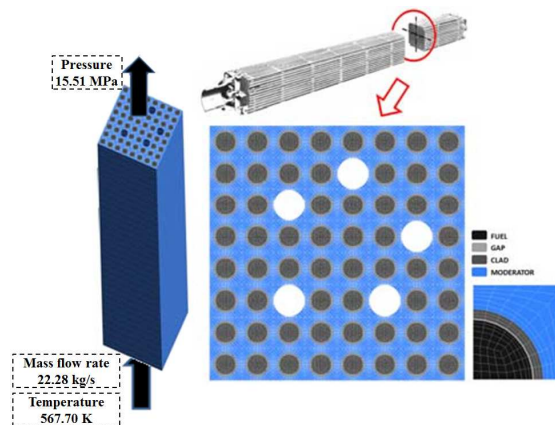


Figure 2: A PWR fuel assembly and a scheme of the channel cross section.

The entire fuel assembly is composed by 16x16 positions and a quarter by 8 x 8: 59 fuel rods and 5 control rods. The fuel rod cladding outer diameter is 10.75 mm. The fuel pellet has a diameter of 9.11 mm and the gap has a thickness of 0.095 mm. The external diameter of the control rod is 13.8 mm. Four different domains have been identified: one fluid, for the moderator, and three different solid domains for fuel, clad and gap. The fuel assembly presents a total of 5 spacers, each 36 mm long, not equally spaced along the length. They are located at the following axial positions: 425 mm, 1062.50 mm, 1593.75 mm, 2231.25 mm and 2762.50 mm. The flow conditions are established with constant pressure at outlet and a mass flow rate with a specified temperature at inlet as shown in Fig. 2.

3.1. Description of the ANSYS CFX Model

Several structured hexahedral meshes created using ICEM CFD were tested in order to set up the computational domain. After a mesh sensitivity analysis the shortest computational time and also independence of the results from the calculation mesh the model with around 1,150,000 elements representing one quarter of a nuclear reactor simplified fuel assembly using symmetric boundary conditions for the axial cut planes. The mesh used in the first development stage [1] [2] has been modified to take into account the presence of spacers. To be able to adequately consider the pressure drop in the spacer a refinement of the axial nodal distribution in the spacer region is needed. The mesh sensitivity analysis shown that around 84,000 elements per spacer are needed to achieve mesh independent results. This geometrical model is used to maintain the computational resources to a relatively low level but to test the coupling methodology with geometry close to reality. In ANSYS CFX the material database has been extended in order to handle the materials used by RELAP5 and to ensure the compatibility between the results. Inlet and outlet boundary conditions used for the CFD simulation are those reported in Fig. 2. A RANS turbulence model, based on the SST $k-\omega$, for the liquid phase is used.

3.1.1. Methodology to consider pressure drop and turbulence enhancement

The methodology proposed to consider the pressure drop and the turbulence produced by the spacer grid consist on includes some source terms in the axial position where spacer grids must be located. A step function will switch from 0 to 1 at spacer grid location in the stream-wise direction.

Pressure drop. The pressure drop along the spacer grid is predicted with a momentum source in the spacer grid area using the mentioned step function. A directional pressure drop model in the streamwise direction is applied. Note that transversal loss and permeability are neglected.

$$S_{M,Z} = -K_{loss}^S \frac{\rho}{2} |U| U_Z \quad (1)$$

A value of 1.8 for the streamwise loss coefficient (K_{loss}^S) has been used. This parameter is empirical adjusted in order to reach the due form loss at spacer and in this case we took the same value used in the RELAP5 simulations to allow results comparison.

Turbulence enhancement. Turbulence produced by the spacer is modeled adding a constant source term (S_k) in the transport equation for turbulent kinetic energy (Eq. 2) in the moderator subdomain. The transport equation shown corresponds to the turbulence model selected for this simulation (SST k- ω).

$$\frac{\partial}{\partial t} \rho k + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_\kappa \frac{\partial \kappa}{\partial x_j} \right] + \tilde{G}_\kappa - Y_\kappa + S_\kappa \quad (2)$$

The value to be used for this source term needs a detailed study. Each spacer grid configuration is different and will generate a different turbulence production profile. The recent released results of the OECD/NEA-KAERI Rod Bundle CFD Benchmark Exercise [4] and the validated CFD model of a rod bundle with spacer grid could [5] be used as a basis for establish the turbulence production more appropriately.

3.2. Description of the PARCS model

A 3D neutronic model of a generic PWR fuel assembly with the characteristics explained above has been created using PARCS 2.7. The cross sections and neutronic parameters for the fuel assembly have been calculated with the SIMTAB methodology based on the joint use of CASMO/SIMULATE [6] [7] [8]. It can handle a wide spectrum of transient conditions since it has been generated for a broad range of temperature and pressure conditions. The method used for the calculation of the Doppler temperature in the fuel is based on the LINC option. The fuel average heat structure temperature is calculated on the basis on the fuel centerline and outer surface temperature calculated by a thermal hydraulic code. The numeric scheme used for calculation is Hybrid. The geometrical description of the fuel assembly model contains 34 nodes in the vertical direction (first node 14 cm, nodes 2 to 33 10.625 cm and last node 20 cm) and 1 in the radial direction both 23 cm long. The boundary conditions are set, as done for full core LWR analysis, as reflective in the x and y directions (neutrons are reflected back into the core). The z boundary conditions for the upper and lower surface of the model are set as zero flux. Each node is assigned to a different planar region. It means that the neutronic parameters in each region are calculated independently to the others.

3.3. Description of the RELAP5 model

In RELAP5 the 1D thermal-hydraulic model of a generic fuel assembly is modeled using the PIPE component connected in parallel to the HEAT STRUCTURE, the solid representing the fuel rods. The PIPE is formed by 34 nodes but only the nodes 2 to 33 represent the actual active length of the fuel assembly, that it the heated fuel rod length. The heat exchange area between PIPE (only all along the active length) and HEAT STRUCTURE is set to be the total external surface of all the fuel rods contained in the fuel assembly. The model considering spacer grids has a form loss coefficient applied at the interfaces where the spacers are located for the calculation of the local pressure loss coefficient is defined to be 1.8 and was empirically adjusted.

4. Results and discussion

The procedure described above to couple the CFD code ANSYS CFX and the neutronic code PARCS has been verified by comparing the results with those produced by the coupled results obtained using RELAP5/PARCS. Both steady state and transient simulations have been performed and the results will be explained in the next sections. Only for the steady state case a simulation of a simplified spacer model to take into account the pressure drop the turbulence enhancement created by them has been used. The wall clock time using 8 INTEL XEON 2000 MHz processors with HPMPI parallel run strategy for a steady calculation was about 55 minutes and 900 minutes for the transient calculation.

4.1. Results of the steady state coupled simulation

A steady state ANSYS CFX/PARCS coupled calculation has been performed. The power distribution (assumed all rods have the same power) along the flow axis and the heat structure temperature profile at $z = 164.69$ cm (Fig. 3) obtained with RELAP5/PARCS match perfectly the ANSYS CFX/PARCS one.

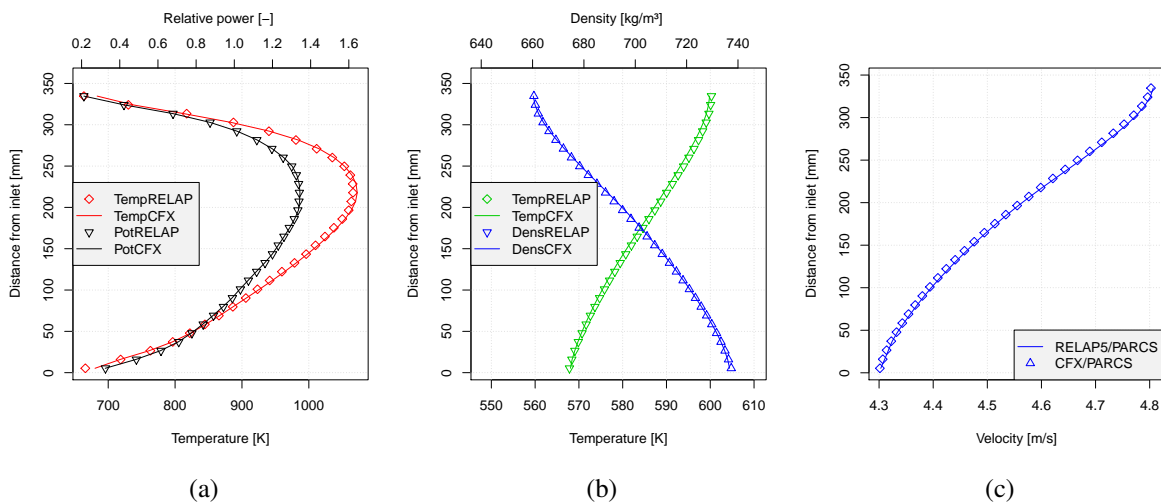


Figure 3: Comparison of the steady state average heat structure temperature and power (a) moderator density and temperature (b) and velocity (c) obtained with ANSYS CFX/PARCS and RELAP5/PARCS

Fig. 4 shows the results of a time dependent moderator inlet temperature changing following a sinus function with an amplitude of 20 K and a period of 20 s. This transient could be representative of operational transient that could take place during a nuclear reactor normal operation. The temperature change is quite limited and the phenomena are taking place with a relatively slow time constant.

4.2. Results of the methodology to predict spacer grids effects

In order to obtain more realistic results by the mean of the CSAP system, the steady state simulation of a ANSYS CFX model considering the simplified spacer grids model have been realized.

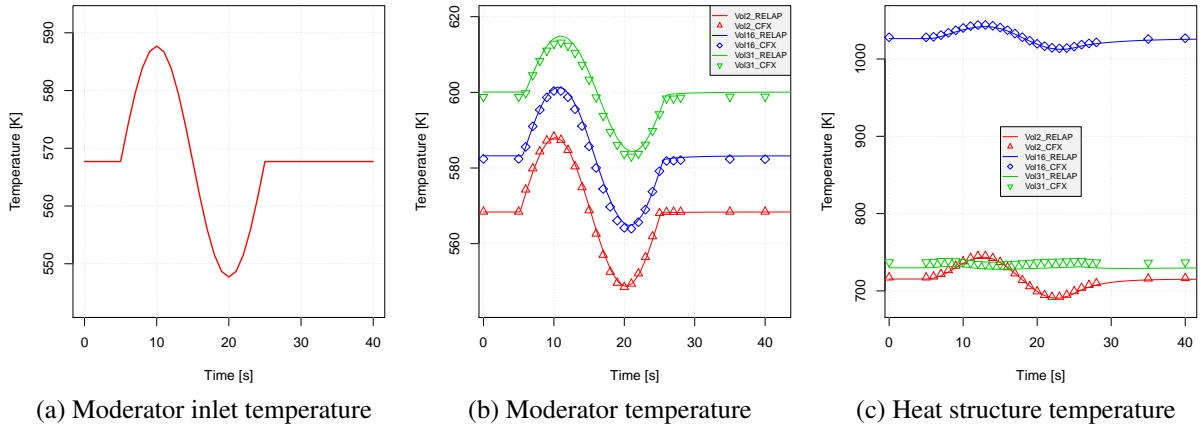


Figure 4: Comparison of the transient average heat structure temperature and moderator temperature at three different axial locations obtained with RELAP5/PARCS and ANSYS CFX/PARCS coupled calculation

In the next sections the pressure drop along the fuel assembly and the turbulence produced by the spacer grids on the channel flow are shown.

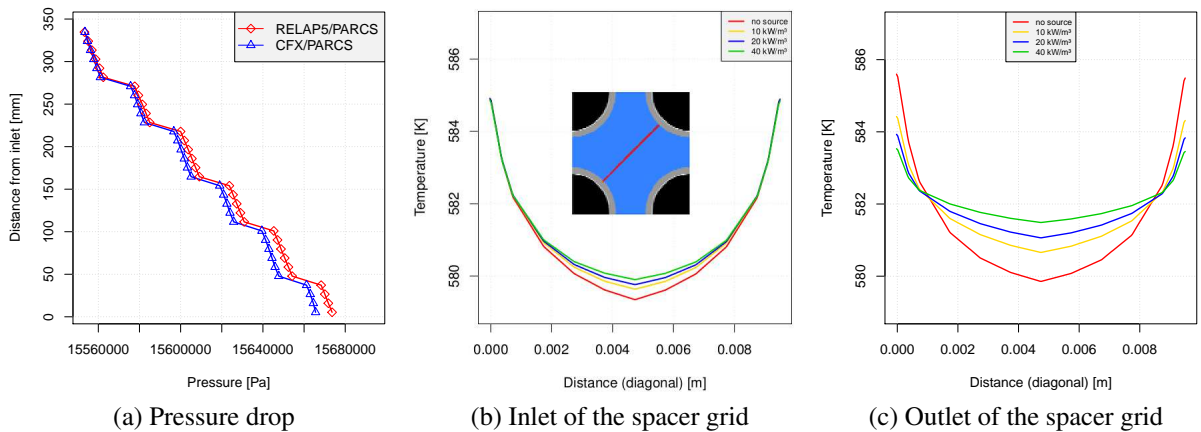


Figure 5: Comparison of the pressure drop obtained with ANSYS CFX and RELAP5/PARCS (a) and temperature distribution at the beginning (b) and final (c) of the spacer grid for different turbulence kinetic energy source terms

4.2.1. Pressure drop prediction

Results of the steady state pressure drop along the channel have been obtained and the pressure result is compared with that obtained by RELAP5/PARCS (Fig. 5a).

It can be noticed that in general the pressure drop predicted by ANSYS CFX/PARCS underestimates the results obtained by the one-dimensional calculation. The discrepancies in the total pressure drop prediction are the combination of the two different components: form and friction pressure losses (note the different curve slope between spacers). A large part of this difference can be explained by the difference between the spatial resolution, 3D in ANSYS CFX and 1D in RELAP, and the way that Navier-Stokes equations are resolved. The difference between the spatial

resolution, 3D in ANSYS CFX and 1D in RELAP, The difference in the pressure drop generated by RELAP5 and the actual CFD model for the single spacer is relatively small but along the whole channel the global difference is given but the sum of them.

4.2.2. Turbulence enhancement prediction

The spacer grid turbulence enhancement effect has been tested applying 4 different constant turbulence kinetic energy volumetric source terms (0 kW/m^3 to 40 kW/m^3). The turbulence kinetic energy obtained in the central spacer ($z=1.313 \text{ m}$) for the 4 cases is shown in Fig 6.

The effect of the turbulence enhancement on the coolant temperature profile is shown in Fig. 5. Here the moderator temperature profiles along the sub-channel diagonal in the central spacer at 1.313 m in the z -direction are shown. Fig. 5b shows the temperature profiles at the spacer grid inlet, Fig. 5c shows the temperature profiles at the spacer outlet for the four cases of Fig. 6.

It can be noticed that the turbulent enhancement influences the flow after the spacer not only in its proximity (right) but also minor residual effect of the different enhancement level are still visible at the next spacer inlet (left). The first main effect of turbulence is that of reducing the clad outer temperature significantly. The reduction is about 2 K out of 5 K wall to bulk temperature difference in the reference case (without applying any turbulence kinetic energy source). The second effect is to flatten the temperature profile obtaining a better peak to average ratio. This is because the turbulent mixing of the flow is stepwise enhanced and lead to a much more effective heat transfer mechanism.

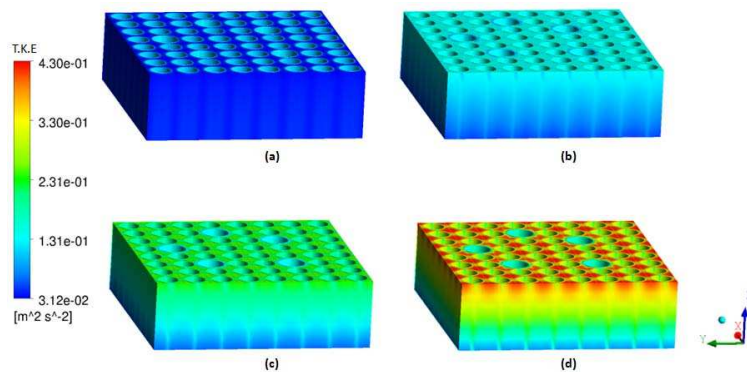


Figure 6: Turbulence Kinetic Energy in the spacer volume. (a) $k = 0 \text{ kW/m}^3$, (b) $k = 10 \text{ kW/m}^3$, (c) $k = 20 \text{ kW/m}^3$, (d) $k = 40 \text{ kW/m}^3$.

5. Conclusions

The generic coupling procedure between the CFD code, in the application shown ANSYS CFX, and the neutron diffusion code PARCS has been presented and tested on a simplified fuel assembly model including a simplified model of spacers grid. The CFD/Neutronic coupling strategy is coherent and leads to very accurate results. This make the study of more realistic transient feasible. The obtained results verify the procedure in both steady state and transient conditions.

Also the attention has been focused on the presentation of a new spacer modeling approach. This lead to an accurate pressure drop calculation along the fuel assembly. Also the effects produced by the spacers generated turbulence on the channel flow are reproduced keeping the computational effort at relatively low level in the same order of magnitude of the original model without spacers. This shown the possibility to reproduce effectively the turbulent mixing effect on the coolant flow created by spacers. For sure, more work has to be done to verify further the simplified spacer model presented in this contribution. Further development is focused on the extension of the coupling code CSAP to make use of the 3D fuel rod power distribution reconstruction module included in PARCS and on the validation of the procedure with larger, more resource intensive models by considering one or more complete fuel assemblies in order to investigate more complex 3D flow-neutronic coupled phenomena. Based on these considerations, it is easy to understand the importance of a simplified spacer grid model to obtain more realistic flow redistribution in the core during asymmetric power distribution transient without requiring much more computational effort.

Experimental correlations and further methodologies will be studied with a validated CFD model [5] to provide more appropriate values and to obtain a heat transfer coefficient properly.

Acknowledgements

This research was supported by the “Plan Nacional de I+D+i” Project EXPERTISER ENE2010-21368-C02-01 and ENE2010-21368-C02-02.

References

- [1] C. Peña-Monferrer, F. Pellacani, S. Chiva, R. Miró, R. Macián-Juan, A Procedure for Coupled CFD-Neutronic Calculation using ANSYS CFX 12.1 and PARCS, in: IV Computational Methods for Coupled Problems in Science and Engineering (Coupled Problems 2011), Kos Island, Greece.
- [2] C. Peña-Monferrer, F. Pellacani, S. Chiva, T. Barrachina, R. Miró, R. Macián-Juan, CFD-Neutronic Coupled Transient Calculation of a Simplified PWR Fuel Assembly using ANSYS CFX 12.1 and PARCS, in: IV Computational Methods for Coupled Problems in Science and Engineering (Coupled Problems 2011), Kos Island, Greece.
- [3] C. Peña-Monferrer, F. Pellacani, S. Chiva, T. Barrachina, R. Miró, R. Macián-Juan, CFD-Neutronic Coupled Calculation of a Quarter of a Simplified PWR Fuel Assembly using ANSYS CFX 12.1 and PARCS, in: 14th International Topical Meeting on Nuclear Reactor Thermal Hydraulics. NURETH-14, Toronto, Ontario, Canada.
- [4] J. R. Lee, J. Kim, C.-H. Song, Synthesis of the OECD/NEA-KAERI rod bundle CFD benchmark exercise, in: Proc. CFD4NRS-4, OECD/NEA & IAEA Workshop, Daejeon, Korea.
- [5] C. Peña-Monferrer, S. Chiva, J. L. M. noz Cobo, E. Vela, F. Pelayo, Participation in the NEA/OECD MATiS-H Benchmark exercise. Study of spacer grids in a rod bundle, in: 38 Reunión Anual de la Sociedad Nuclear Española, Cáceres, Spain.
- [6] R. Miró, T. Barrachina, F. Maggini, O. Rosello, G. Verdú, A. Gómez, A. Ortego, J. C. Martínez-Murillo, Utilization of SIMTAB methodology in translating the kinetics parameters from SIMULATE-3 to RELAP5/PARCS for REA 3d-dynamic analysis in Trillo NPP, in: ENS International Meeting on LWR Fuel Performance, Salamanca, Spain.
- [7] O. Roselló, Desarrollo de una metodología de generación de secciones eficaces para la simplificación del núcleo de reactores de agua ligera y aplicación en códigos acoplados neutrónicos termohidráulicos, 2004.
- [8] J. Cronin, K. Smith, D. V. Planck, SIMULATE-3. Advanced three-dimensional two-group reactor analysis code, Studsvik/SOA-95/18.