*2011 International Nuclear Atlantic Conference - INAC 2011*
*Belo Horizonte,MG, Brazil, October 24-28, 2011*
*ASSOCIAÇÃO BRASILEIRA DE ENERGIA NUCLEAR - ABEN*
***ISBN:*** **978-85-99141-04-5**

# CFD-NEUTRONIC COUPLED CALCULATION OF A QUARTER OF A SIMPLIFIED PWR FUEL ASSEMBLY INCLUDING SPACER PRESSURE DROP AND TURBULENCE ENHANCEMENT

**C. Peña[1], F. Pellacani[1], S. Chiva[2,*], T.Barrachina[3], R. Miró[3] and R. Macián Juan[1]**

[1] Ntech Lehrstuhl für Nukleartechnik
Technische Universität München
Boltzmannstr. 15
85748 Garching, Germany
carlos.pena@ntech.mw.tum.de, pellacani@ntech.mw.tum.de, macian@ntech.mw.tum.de

[2] Departamento de Ingeniería Mecánica y Construcción
Universitat Jaume I
Campus del Riu Sec
12080 Castellón de la Plana, Spain
schiva@emc.uji.es

[3]Institute for Industrial, Radiophysical and Environmental Safety (ISIRYM)
Universitat Politècnica de València
Camí de Vera, s/n
46022 Valencia, Spain
rmiro@iqn.upv.es, tbarrachina@iqn.upv.es

## ABSTRACT

A computational code system 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. This work represents one of the first attempts to couple the multiphysics of a nuclear reactor core with a 3D spatial resolution in a computer code.
This will open new possibilities regarding the analysis of fuel elements, contributing to a better understanding and design of the heat transfer process and specific fluid dynamics phenomena such as cross flow among fuel elements. The transient simulation of control rod insertion, boron dilution and cold water injection will be made possible with a degree of accuracy not achievable with current methodologies based on the use of system and/or subchannel codes.
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 subroutines 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.

*Corresponding author

# 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 Dynamic (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.

This paper describes a coupling procedure between a *generic* CFD code and a neutron diffusion code and its application to ANSYS CFX 13.0, a commercial CFD code and to the 3D transient neutron diffusion program PARCS. The resulting coupled code system, called Coupled Solver ANSYS-CFX/PARCS (CSAP), has been verified with a steady state test considering a quarter of a fuel assembly subchannel. The results have been compared with those obtained using the coupled code system RELAP5/PARCS to evaluate the consistency and correctness of the procedure.

Nuclear fuel bundles contain spacers essentially for mechanical stability and to influence the flow dynamic and heat transfer phenomena along the fuel rods. In April 2011 a blind OECD/NEA benchmark was started. It is called “Turbulent Flow in a Rod Bundle with Spacers”. Detailed data of spacers have been made available to the international community. In the framework of this benchmark a 5x5 fuel fuel bundle with spacer will be investigated in order to assess what are the best modelling options to capture the essential features of the turbulent structures downstream of the spacer, and the heat transfer mechanism.

Given the complexity of these components, an analytical approach required the detailed knowledge of the geometrical data. The use of a refined calculation mesh in the spacer region and its vicinity is of primary importance. The simulation of bigger domains, like one or more fuel assemblies would lead to an extremely high computational effort. In fact, the complexity of the global model would affect the calculation time that we want to keep low in this first phase of development.

Based on these considerations, a method to reproduce the effect on both, fluid dynamics and

heat transfer, of spacer grids without their detailed reproduction have been investigated.
The method consists in implementing pressure drop and turbulence enhancement effects by the mean of volumetric sources applied to subdomains positioned where the spacers are located.

## 2. DESCRIPTION OF THE COUPLED AND THE DATA EXCHANGE PROCESS

This chapter 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. Furthermore the data exchange and the synchronization processes are explained in detail. 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. Ferrando et al. [1] and Conner et al. [2] have assessed CFD codes, at different resolution level, for nuclear safety applications. 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. Fig. 2.1 shows an overview of the parameters needed by the two codes to perform a coupled simulation.

The coupling the CFD code ANSYS CFX to the neutron diffusion code PARCS makes it feasible to obtain results with a higher resolution regarding the local thermal hydraulic behavior of the fuel. The power produced in the nuclear fuel is dynamically calculated by the solution of the neutron diffusion equation by PARCS depending on the *local* and *instantaneous* conditions calculated by the CFD code appropriately adjusted for the neutronic nodal mesh size. PARCS provides the power distribution in each fuel rod, and this information is used by ANSYS-CFX as heat sources in the solid elements that simulate the individual fuel rods of the fuel assembly. A conjugate-heat transfer solution developed by the authors with detailed radial and axial nodalization of individual fuel rods is then coupled to the general fluid flow ANSYS-CFX solution in the flow regions surrounding the fuel rods, and the new flow field is calculated. This is done for each time step in a transient or iteration

step in a steady state solution convergence criteria and time step sizes are selected according to the characteristics of the neutronic and CFD solution at each time or iteration step, so that the more restrictive ones from the point of view of accuracy and stability are selected for both codes. In our experience, the CFD solution is usually the one controlling the calculation process.

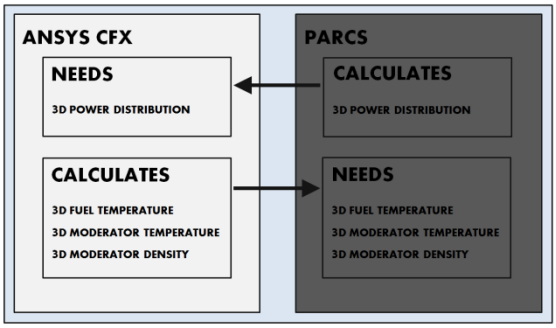


**Figure 2.1. Overview of the parameters needed by the two codes to perform a basic coupled simulation.**

The CSAP coupling procedure and the first preliminary results are explained in detail in previous contributions [3] [4]. In this work only the working principle and latter improvements are reported. Fig. 2.2 describes the data exchange flow.

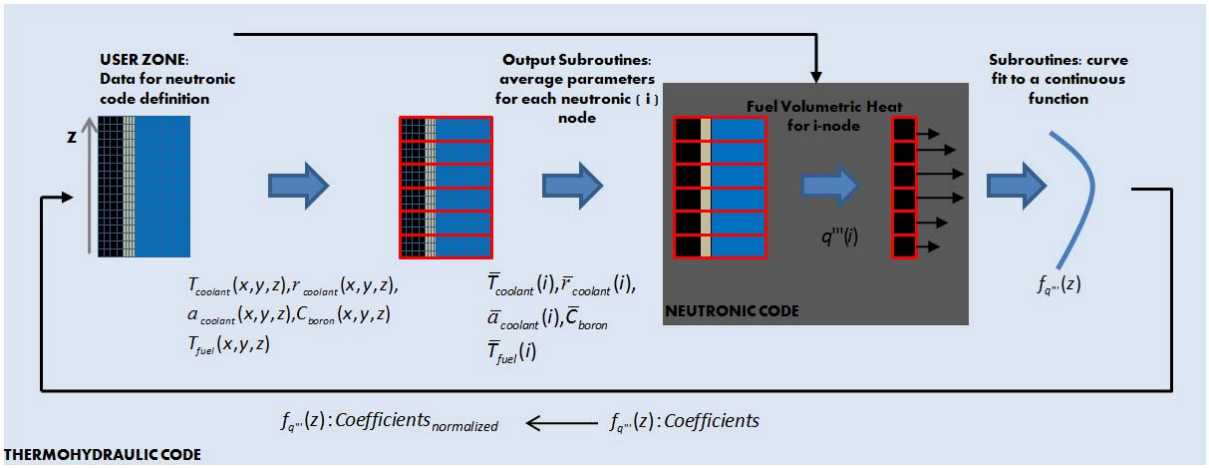


**Figure 2.2. Data exchange flow.**

The last version of the dynamic library CSAP contains 5 subroutines with different tasks: mapping, calculation of the averaged values needed by the neutronic calculation, data exchange and synchronization by means of PVM calls and interpolation of neutronic data needed by the CFD calculation.

The developed subroutines are contained in a dynamic library (the current version is CSAP 1.1). The tasks carried out by each of them are summarized next. In Fig. 2.3 a flow diagram representing the coupling scheme and scheme of the communication between processes is shown. The coupling scheme is purely *explicit*. The thermal-hydraulic data calculated by ANSYS CFX and elaborated by CSAP are used for the calculation of the power distribution in PARCS at iteration *n*. This calculated heat distribution is then used in the thermal-hydraulic calculation at iteration *n+1*.

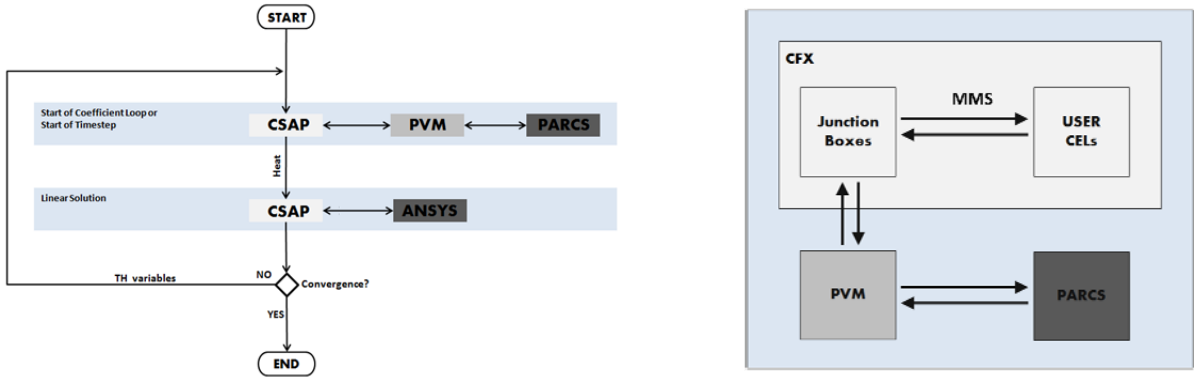


**Figure 2.3. Explicit coupling scheme flow diagram (left) and schematic representation of the communication between processes (right).**

## 3. DESCRIPTION OF THE MODELS

This chapter describes the problem to be solved by the coupled tools and the way in which the geometrical model has been implemented in the CFD and neutronic code. A quarter of the nuclear reactor fuel assembly will be examined; a simplified sketch of the model cross section is shown in Fig. 3.1.

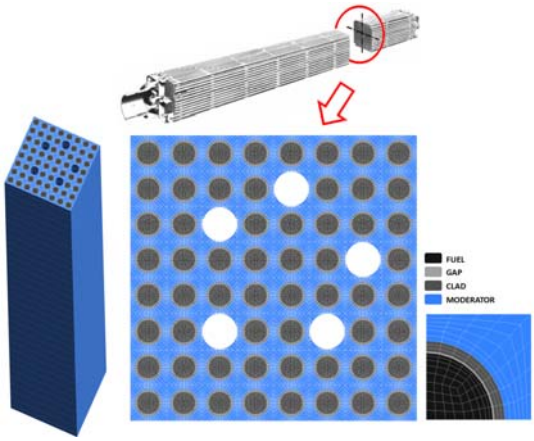


**Figure 3.1. A PWR fuel assembly and a scheme of the channel cross section.**

The entire fuel assembly is composed by 16x16 positions while 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 boundary conditions are summarized in Table 2.1.

**Table 2.1: Boundary conditions for the steady state calculation**

| | |
|---|---|
| Moderator Inlet Temperature | 567.7 K |
| Moderator Mass Flow Rate | 22.277 kg/s |
| Reference Pressure | 15.51 MPa |
| Average power of FA at full power | 17.05 kW |

## 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 independency of the results from the calculation mesh the model with by ca. 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 [3],[4] has been modified to take into account the presence of spacers. To be able to adequately the pressure drop in the spacer an refinement of the axial nodal distribution in the spacer region is needed. The mesh sensitivity analysis shown that ca. 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.

The model consists of four different domains: one fluid and three solid domains (see Fig. 3.1). All of them are connected together by interfaces with the heat transfer option “Conservative Heat Flux”. Heat is only generated in the fuel domain but is transferred to the fluid through heat conduction in the gap and in the clad regions. The gap is modeled based on the gap conductance model. Heat transfer in the gap is in reality taking place based on different mechanisms: heat conduction, solid contact if the fuel pellet is cracked and restructured and radiation. Usually convection is neglected due to the conditions of the gas mixture contained in the fuel rod. For this reasons since approximation is required, the material contained in the gap has been defined in a solid state with characteristics of conductivity and density based on those included in the manual of RELAP5 to ensure the comparison between the results. 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. The boundary conditions used for the CFD simulation are those reported in Table 2.1. A RANS turbulence model, based on the SST, for the liquid phase is used.

The pressure drop in the spacers is predicted applying a momentum source along the spacers length. This has been implemented using the «Momentum Directional Loss Model» provided by ANSYS. Since the pressure drop in the transverse directions and the permeability are not considered the momentum loss in the streamwise direction (Z axis) is:

$$S_M = -K_{loss}^S (\rho / 2) |U| U_z \tag{1}$$

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.

Furthermore, the turbulence produced by the spacer is modeled adding a constant source per unit volume for the quantity k (Turbulence Kinetic Energy) in the spacer subdomains. The value to be used for this source coefficient need a detailed study. Each spacer configuration is different and will generate a different turbulence enhancement One of the expected goal of the OECD/NEA Benchmark previously referenced is that of provide such values.

### 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 and developed at the Polytechnic University of Valencia together with Iberdrola [5] [6] [7]. 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. No fuel spacers have been considered, and only friction pressure losses along the fuel rods contribute to the pressure drop in the fuel assembly.

The boundary conditions applied are summarized in Table 2.1. In this way, it is possible to use a relatively simple modeling strategy to represent in an effective way even complicated geometries. The 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 OF SIMULATION

The procedure described above to couple the CFD code ANSYS CFX and the neutronic code PARCS has been validated 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 a simplified spacer model to take into account the pressure drop the turbulence enhancement created by them has been used.

Partitioning strategy plays an important role not only to minimize the communication time while maintaining the load balance. In this special case, it is playing an important role to obtain acceptable results by the means of user develop subroutines.

An analysis of the wall clock time for a steady state calculation has been performed in function of the number of the processors used for calculation [5]. The results of this analysis is that the computational time reduces exponentially from 2 to 8 processors. The number of 8 processors resulted to be the optimum for our domain and the time saving increment given by more than 8 processors is relatively low. 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 both transient calculations.

### 4.1. Results of the Steady State Coupled Simulations

A steady state ANSYS CFX/PARCS coupled calculation has been performed. The power distribution (assumed all rods have the same power) (Fig. 4.1 left) along the flow axis and the radial heat structure temperature profile at $z$ = 164.69 cm (Fig. 4.1 right) obtained with RELAP5/PARCS match perfectly the ANSYS CFX/PARCS one.

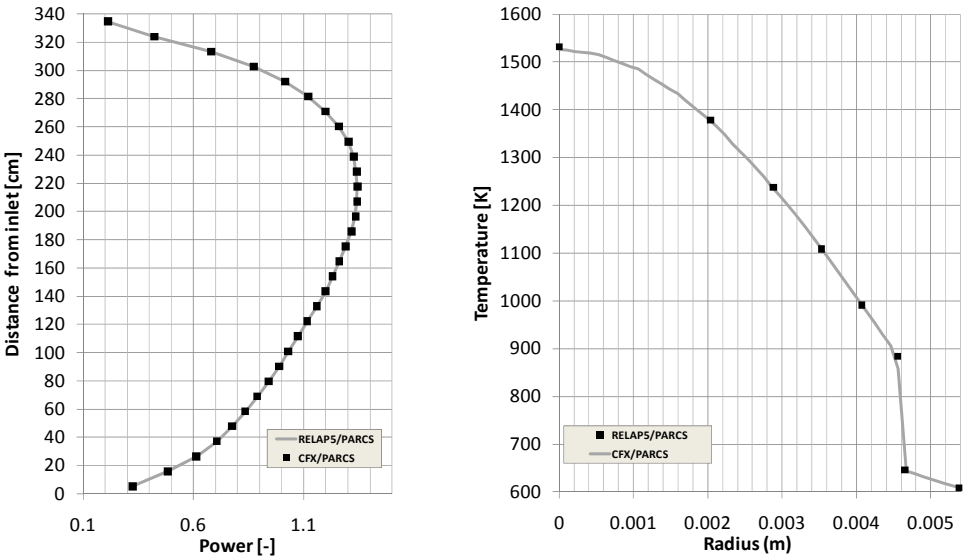


**Figure 4.1. Comparison of the steady state power distributions (left) and heat structure radial temperature profile at $z$ = 164.69 cm(right).**

Also the average fuel centerline temperature (Fig. 4.2 left) and the average fuel outer surface along the flow axis (Fig. 4.2 right) obtained with RELAP5/PARCS match perfectly the ANSYS CFX/PARCS one.

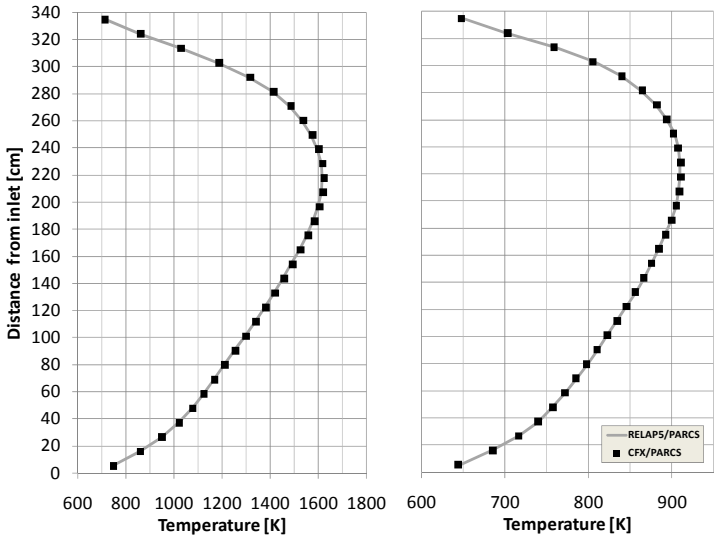


**Figure 4.2. Comparison of the steady state fuel centerline temperature (left) and fuel outer surface temperature (right).**

The average moderator temperature, moderator density and velocity along the flow axis are shown in the diagrams of Fig. 4.3. The values of the ANSYS CFX/PARCS coupled calculation are volume averages for those computational volumes that are contained in one of the much coarser computational volumes of RELAP5. The results show a near perfect agreement between both coupled solutions, thus validating the coupling procedure developed in CSAP. It is important to note that these results do not show all the capabilities of the ANSYS CFX/PARCS coupled code system, since detailed local values for flow and fuel rod variables have not been plotted, and the problem chosen does not contain intra fuel assembly asymmetries: all fuel rods have the same power. The choice of problem was dictated by the capabilities available in RELAP5-PARCS, so that both simulations could be rigorously compared.

The k-effective calculated by the RELAP5/PARCS simulation is 1.054313 and that calculated by ANSYS CFX/PARCS is 1.054412. The reactivity difference is about 9.39E-10 pcm and can be considered acceptable.

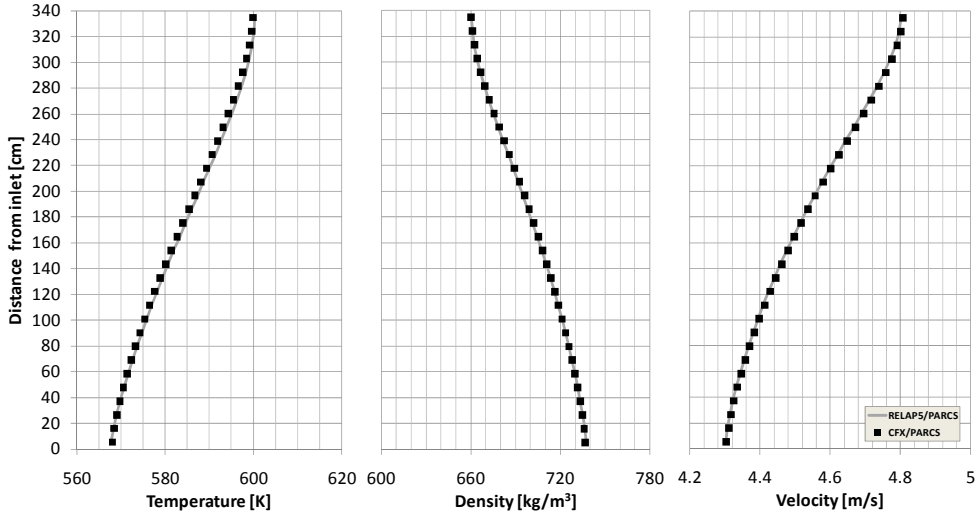


**Figure 4.3. Comparison of the steady state average moderator temperature (center), moderator density (center) and moderator velocity (right) obtained with RELAP5/PARCS and ANSYS CFX/PARCS coupled calculation.**

## 4.2. Results of Transient Coupled Simulation

Two different transient simulations have been realized and the results are shown in Fig. 4.4 and 4.5.

### 4.2.1. Results of "sinus" transient simulation

Fig. 4.4 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.

The results obtained with ANSYS CFX/PARCS and with RELAP5/PARCS are compared at three different axial locations: node 2 node 16 and node 31 of the RELAP5 HEAT STRUCTURE nodalization. The average transient heat structure temperature (Fig. 4.4 upper) and the average transient moderator temperature (Fig. 4.4 lower) obtained with RELAP5/PARCS match perfectly the ANSYS CFX/PARCS one.

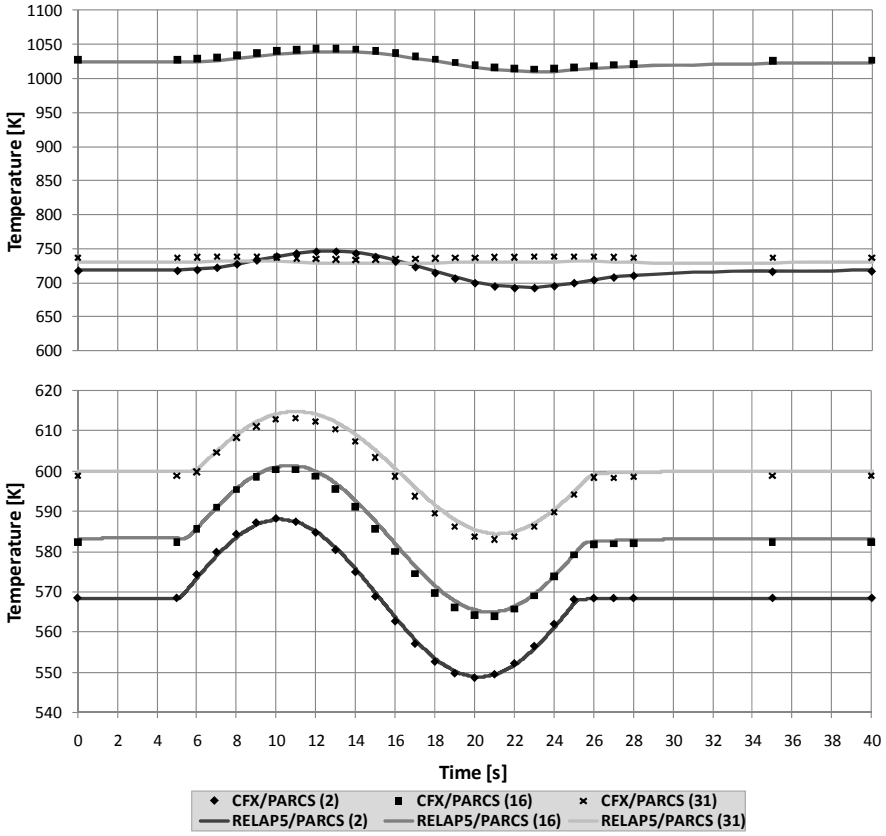


**Figure 4.4. Comparison of the transient average heat structure temperature (upper) and moderator temperature (lower) at three different axial locations obtained with RELAP5/PARCS and ANSYS CFX/PARCS coupled calculation – Sinusoidal Inlet moderator temperature.**

### 4.2.2. Results of the “step” transient simulation

Fig. 4.5 shows the results of a time dependent moderator inlet temperature changing following a step function. The inlet water temperature suddenly decreases of 50 K.
This transient could be representative of an accidental condition where safety related mechanisms are taking place and suddenly water is injected at a lower temperature to help the cooling down of the reactor. In this case the phenomena time constants are smaller than the previous simulation.

The results obtained with ANSYS CFX/PARCS and with RELAP5/PARCS are again compared at the same three different axial locations. The average transient heat structure temperature (Fig. 4.5 upper) and the average transient moderator temperature (Fig. 4.5. lower) obtained with RELAP5/PARCS match perfectly the ANSYS CFX/PARCS one.

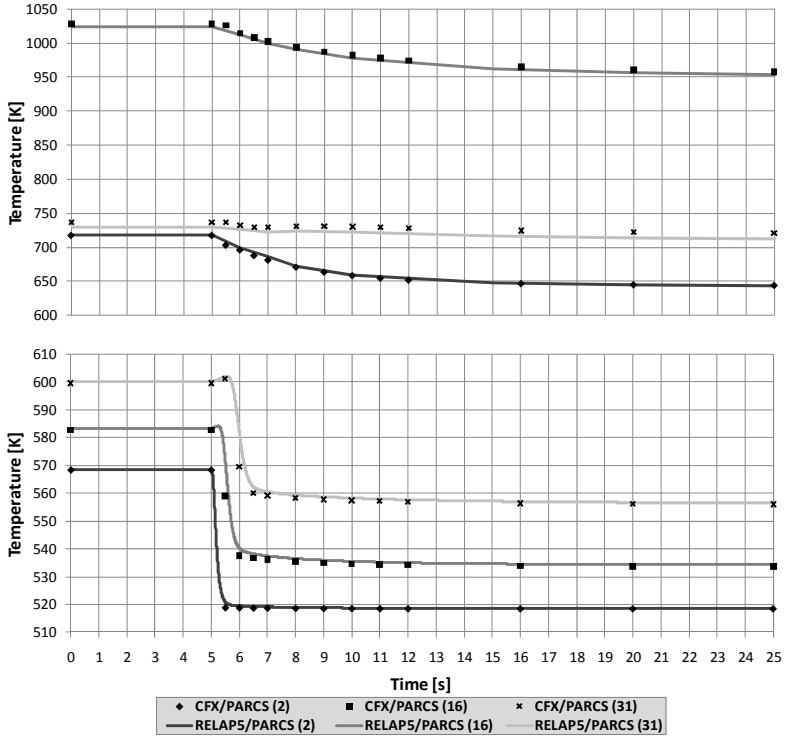


**Figure 4.5. Comparison of the transient average heat structure temperature (upper) and moderator temperature (lower) at three different axial locations obtained with RELAP5/PARCS and ANSYS CFX/PARCS coupled calculation - Inlet moderator temperature step (-50K).**

## 4.3. Results of the Steady State Coupled Simulations Including a Simplified Model for the Spacers

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. In the next sections two different results are shown. The pressure drop along the fuel assembly and the effects produced by the spacers generated turbulence on the channel flow.

### 4.3.1. Pressure drop prediction

Results of the steady state pressure drop along the channel have been obtained and the absolute pressure result is compared with that obtained by RELAP5/PARCS in the Fig. 4.6.

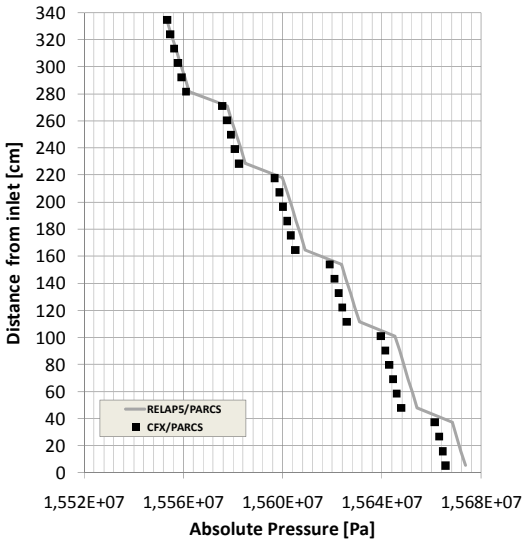


**Figure 4.6. Comparison of the pressure drop obtained with RELAP5/PARCS and ANSYS CFX/PARCS coupled calculation.**

It can be noticed that in general the pressure drop predicted by ANSYS CFX/PARCS in general 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). The difference in the pressure drop generated by RELAP 5 and the actual CFD model for the single spacer is relatively small but along the whole channel the global error is given but the sum of them. For this reason the error obtained by each spacer needs to be further reduced and improvements still need to be done. In order to create a more realistic model that could take into account also the flow area reduction in the spacer, a porous media model could be introduced. This would increase coolant velocity in the spacer region and lead to a more realistic pressure drop calculation.

### 4.3.2. Turbulence enhancement prediction

The spacers 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 m) for the 4 cases is shown in fig. 4.5.

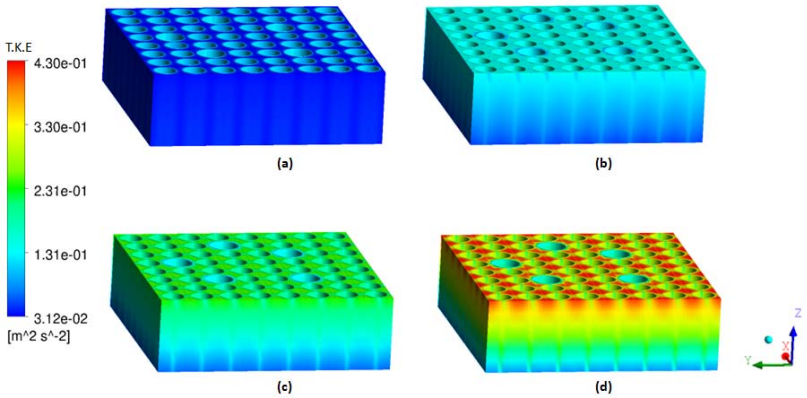


**Figure 4.7. Turbulence Kinetic Energy in the spacer. (a) k= 0 kW/m$^3$, (b) k= 10 kW/m$^3$, (c) k= 20 kW/m$^3$, (d) k= 40 kW/m$^3$.**

The effect of the turbulence enhancement on the coolant temperature profile is shown in figure 4.8. Here the moderator temperature profiles along the subchannel diagonal in the central spacer at z=1.313 m are shown. Fig. 4.8 left shows the temperature profiles at the spacer inlet, figure 4.8 right the shows the temperature profiles at the spacer outlet for the four cases of figure 4.7.

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.

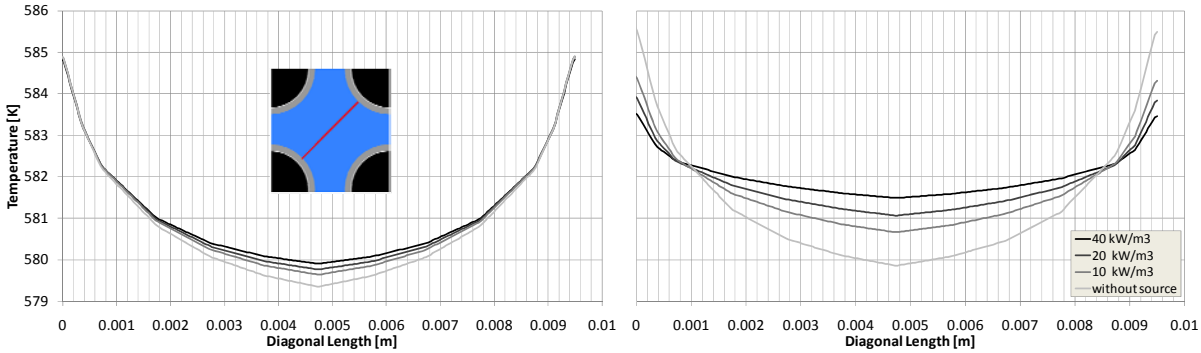


**Figure 4.8. Temperature distribution at the start of the spacer (left) and at the final (right) for different turbulence kinetic ener gy sources.**

## 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 validate the procedure in both steady state and transient conditions.

Also the attention has been posed in 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 realitively low level in the same order of magnitude of the original model without spacers.Several turbulent enhancement level have been considered. 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 validate 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.

## ACKNOWLEDGMENTS

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

## REFERENCES

1. J. Ferrando, R. Miró, S. Chiva, G. Verdú, “Detailed CFD simulation of a PWR fuel rod and fuel element Mathematics and Computation”, Supercomputing, Reactor Physics and Nuclear and Biological Applications, September 12-15, 2005, Avignon, France.
2. Michael E. Conner, Emilio Baglietto, Abdelaziz M. Elmahdi, “CFD methodology and validation for single-phase flow in PWR fuel assemblies”, *Nuclear Engineering and Design*, **Vol. 240**, Issue 9, pp. 2088-2095 (2010).
3. *C. Peña, S. Chiva, R. Miró, F. Pellacani and R. Macián,* “A Procedure for Coupled CFD-Neutronic Calculation using ANSYS CFX 12.1 and PARCS”, Computational Methods for Coupled Problems in Science and Engineering 2011, Kos, Greece.
4. *C. Peña, S. Chiva, R. Miró, T. Barrachina, F. Pellacani and R. Macián,* “CFD-Neutronic Coupled Transient Calculation of a Simplified PWR Fuel Assembly using ANSYS CFX 12.1 and PARCS”, Computational Methods for Coupled Problems in Science and Engineering 2011, Kos, Greece.
5. R. Miró, T. Barrachina, F. Maggini, O. Roselló, 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.”, ENS International Meeting on LWR Fuel Performance, 2006 Salamanca Spain.
6. 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”, PhD Thesis, UPV, 2004.
7. J.T. Cronin, K.S. Smith, D.M. Ver Planck, “SIMULATE-3. Advanced three-dimensional two-group reactor analysis code”, Studsvik/SOA-95/18, 1995.