Asociación Argentina de Mecánica Computacional

Mecánica Computacional Vol XXXIII, págs. 2845-2855 (artículo completo)
Graciela Bertolino, Mariano Cantero, Mario Storti y Federico Teruel (Eds.)
San Carlos de Bariloche, 23-26 Setiembre 2014

CALCULATION OF PRESSURE DROP IN A PRELIMINARY DESIGN OF NUCLEAR FUEL SPACER GRIDS IN AN INTEGRAL PWR USING CFD METHODS

J.D. Jiménez Díaz, A.I. Lazarte and J.C. Ferreri[a]

jjimenez@arn.gob.ar alazarte@arn.gob.ar jferreri@arn.gob.ar

Autoridad Regulatoria Nuclear (ARN), Av. Del Libertador 8250, CABA, BNP 1429C, Argentina

Keywords: Computational Fluid Dynamics, nuclear fuel bundles, spacer grids, pressure loss

Abstract. This work carries out a calculation of the pressure drop of the coolant flow past spacer grids in a preliminary design of a nuclear fuel assembly of a modular, integral PWR. In this small modular, integral reactor the coolant flows along the core driven by natural circulation. The analysis will focus on considering a cross section of 1/12 of the entire fuel element despite a single asymmetry and an axial segment. A 3D CFD simulation is performed to estimate the pressure drop during steady state flow rate of single-phase light water at constant temperature. Bundle cross-flows are disregarded as a first approximation. Appropriate boundary conditions are applied at fuel pin walls and symmetry planes, namely outlet absolute pressure and mass flow rate at inlet that are kept constant. Results presented in non-dimensional, normalized way show the expected behavior. However, due to modelling hypothesis based on a limited knowledge of spring geometrical details, the results cannot be considered useful for design optimization purposes.

Work done in the framework of the Collaboration Agreement between ESSS Argentina and J.C.Ferreri (through Academia Nacional de Ciencias de Buenos Aires), March 2014. ESSS Argentina made available the full package of ANSYS CFX-FLUENT© v-15 for this academic use.

[a] *Emeritus advisor*, alternate e-mail: *jcferreri@gmail.com*

1. INTRODUCTION

The use of computational fluid dynamics (CFD) codes to optimize preliminary or conceptual designs of nuclear components increased dramatically in the last years. This type of simulation approximation is a complement to usual, accredited engineering design tools, corroborated by experiments. However, detailed calculations using CFD permit a degree of analysis in depth of minute design changes and investigations of flow pattern details that are in many cases only available through complicated experimentation. In this way, the efforts and costs of experimentation may reduce to the study of almost definitive prototypes for fuel designs.

Fuel elements are a key component of any type of nuclear installation or power plant using nuclear fuel bundles. In the case of coolant flow driven by natural circulation, like in advanced nuclear or innovative reactors designs, the use of CFD codes to evaluate concentrated pressure losses due to fuel elements spacer grids becomes relevant. This is due to the importance of the minimization of pressure losses in the primary flow circuit because of the limited driving force. An appropriate sequence for such analyses may be, namely: a) setup of a preliminary design. b) Detailed flow analysis using a CFD approximation. c) Theoretical optimization of a prototype. d) Setup of an experimental study, implementation of an experimental rig and obtainment of results. e) Final design adjustment and f) finally and more importantly, validation of the simulation by obtainment of new, hopefully improved, results.

Several examples may illustrate the importance of the subject of this paper. The International Topical Meetings on Nuclear Reactor Thermal-hydraulics (NURETH), denoted as N15 and N14 in the references list, are appropriate fora to test how timely the problem is. The following references illustrate a non-exhaustive list of papers directly related to the present one: Campagnole dos Santos et al. (2013), Caviezel et al. (2013), Conner et al. (2011), Frank et al., (2011), Karoutas et al. (2013), Kim et al. (2011), Krepper and Rzehak (2013), Lascar et al. (2013), Petrov et al. (2013), Tóth et al. (2013) and Yan et al. (2011). In addition to the above, benchmarks on CFD code capabilities on the subject have been held. Smith et al. (2011) and Périn et al. (2013) show relevant reports on this type of activities. Additionally to these reports, there are specific meetings dealing with the verification and validation of computer codes. The CFDNRS-4 Meeting (named as CFD4 in the references list) held in 2012 is a particular good source for this activity. Some examples of this information, related to a benchmark exercise, are Chang et al. (2012) and Lee et al. (2012). Specific reports, again directly related to this paper, are Melideo et al. (2012), Barthel et al. (2012), Frank et al. (2012), Yan et al. (2012), Krepper et al. (2012) and Yudov (2012). These reports give valuable details on the status of the experimental work and on the validity of simulations and, more specifically, on physical models used, grid size effects and convergence of results.

This paper presents a set of results related to a hypothetical design of a spacer grid, aimed at gaining experience in advance to regulatory analysis requirements for CFD approximations to coolant flow along specific components of nuclear power plants. This practice is somewhat recent in Argentina and some examples supporting nuclear safety analysis may be found in Lencina and Ballesteros (2013). The fuel element design considered in the present paper design is conceptually coherent with the ones used in small modular reactors like CAREM25, see e.g. is Boado et al. (2011). Calculations of single pressure drops like the one imposed by spacer grids may be predicted with reasonable accuracy, without looking for a very detailed analysis of the complex flow patterns produced by mixing vanes. Since this conceptual design relates to natural circulation driven primary coolant flow, the usual mixing vanes of typical

PWR or BWR spacer grids are not present. This condition is an advantage for this study because it allows the use of a somewhat low-resolution grid. The results presented in the following sections are consistent with these approximations.

2. APPROXIMATE DESIGN GEOMETRY OF A SPACER GRID

The geometry considered conceptually corresponds to the CAREM reactor. This reactor design has been described in many sources and one describing the whole reactor and its subsystems is Boado et al. (2011). Figure 1 shows a global view of the fuel element and Figure 2 depicts a cross section of the fuel assembly and guide tubes for the control rods. The fuel element consists of 108 fuel rods, 18 control rod channels and an instrumentation tube. Four spacer grids are located equidistantly, avoiding fuel rods transverse movement. The fuel assembly components are typical of PWR designs. The active length of the fuel rods is 1.4 m.

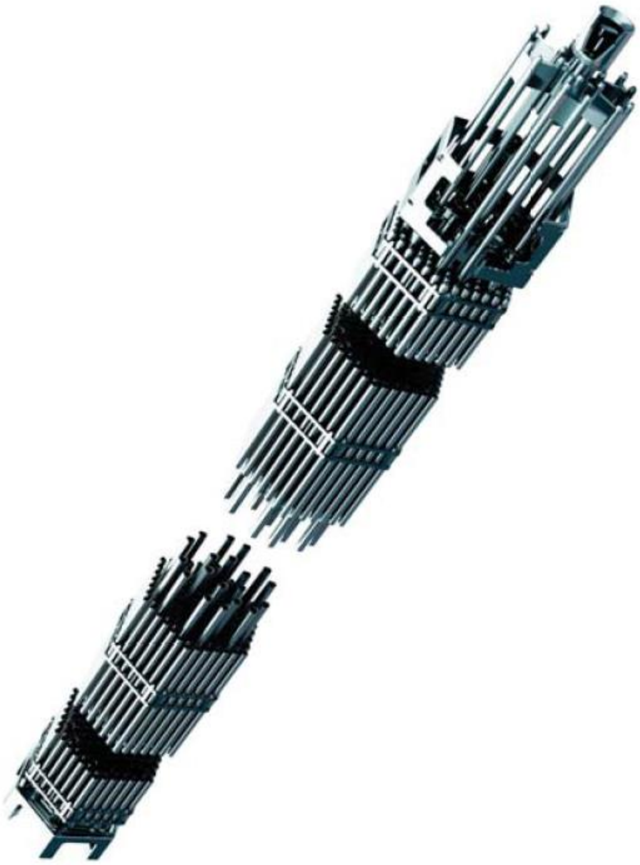

Figure 1 Schematic view of the fuel assembly, from Boado et al. (2011).

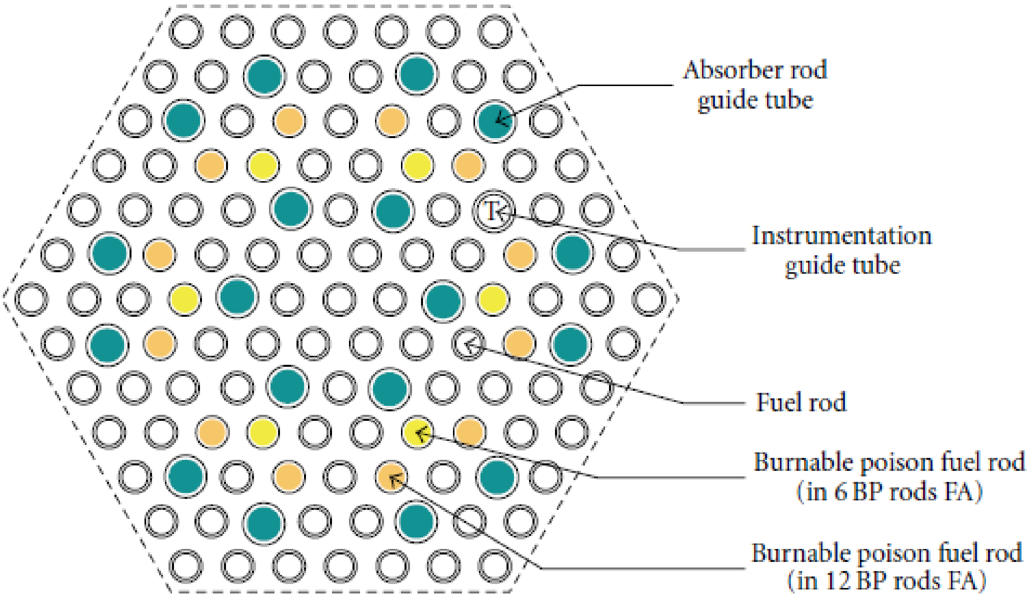

Figure 2 Cross section of the fuel assembly and guide tubes, from Boado et al. (2011).

From the hydraulic point of view, the assembly shows only one asymmetry, corresponding to an instrumentation tube. Due to this and to the guessed geometrical configuration to perform the present study, this lack of symmetry will be disregarded in what follows, so the analysis considers one-twelfth of the cross section illustrated in Figure 2. This reduced cross section can be seen in Figure 3 that comes from an approximation of a typical grid spacer. It is composed of hexagonal flow passages with two fixed supports for the rods and a space to be filled with some elastic support to damp their flow induced transverse vibrations. An approximate geometrical representation of the elastic support will be considered later on. The partial channel includes one guide tube from the external ring, a half of a guide tube from the internal ring, five fuel rods, eight half fuel rods and one-twelfth of the central fuel rod. The center converging lines represent the planes of symmetry and the domain is closed with a section of the outer assembly wall.

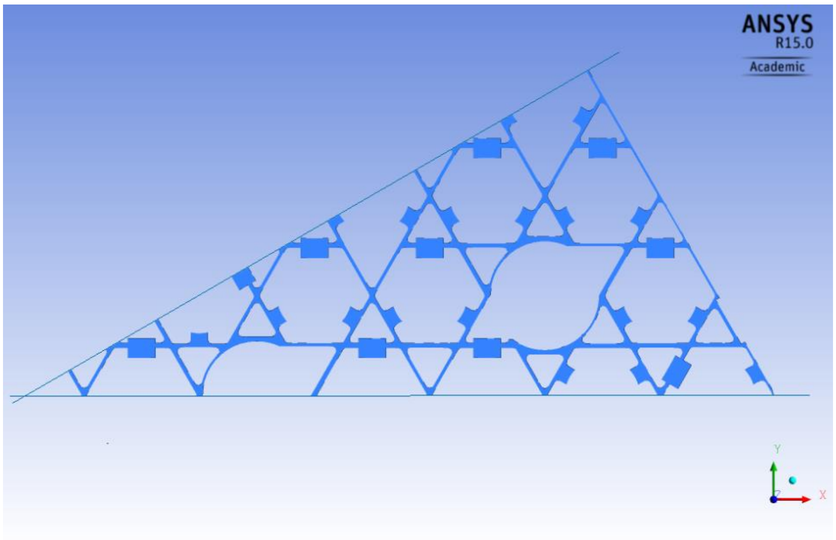

Figure 3 Cross section of the spacer grid considered in the present analysis.

Figure 4 is a 3D representation of this hypothetical spacer grid. The springs that must be provided have not been represented because of the lack of a specific design but it is postulated that a restriction to flow equivalent to a fraction of the fixed support may be supplied. This restriction will be placed in the same plane as the fixed supports with sharp edge facing the flow. Eventually, this contribution may be subject to a sensitivity analysis. These flow restrictions are shown in Figure 4.

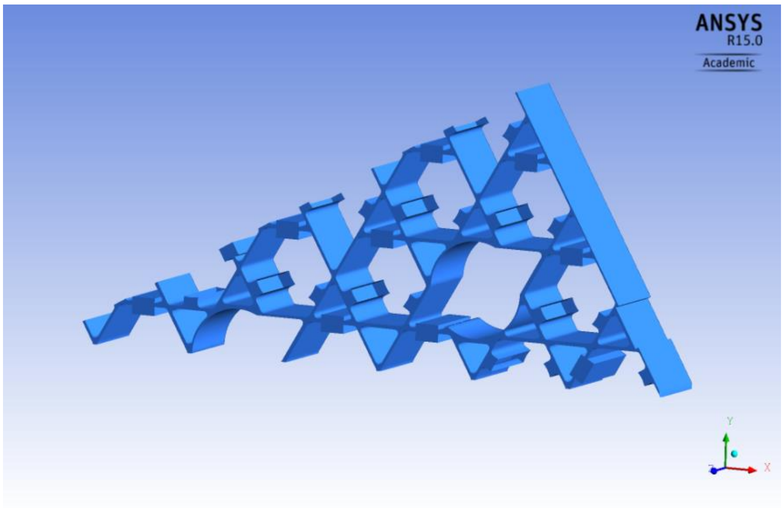

Figure 4 A design of a spacer grid suitable for natural circulation flow along fuel bundles.

Figure 5 shows a segment of the assembly conceptually shown in Figure 1 and its total length is 0.61 m. The fuel pins, the guide tubes and the spacer grid are partially shown in order to exemplify the integration domain that will be subject to analysis, considering its different components. The flow domain considers the spacer grid in the middle. The length considered is enough to get linear distribution of pressure loss along the fuel channel and fully develop flow pattern.

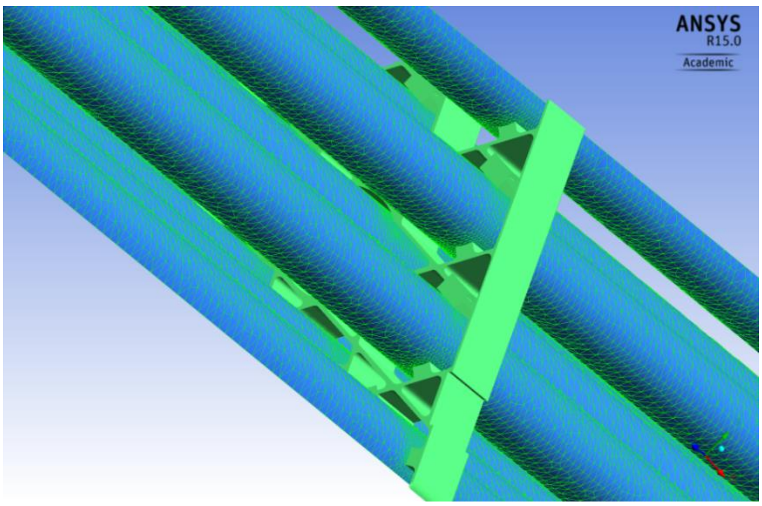

Figure 5 A 3D view of the fuel bundle components showing some details of the discrete mesh on the fuel pins surfaces.

3. CFD APPROXIMATION

Simulations have been performed using ANSYS CFX-15 (ANSYS, 2013) Academic version, managed through ANSYS Workbench. The governing equations for this simplified analysis considering 3D flow of isothermal, single phase light water at 343.16 K are well established and may be found in the code documentation. The fuel assembly is in vertical position. Working pressure was 1400 kPa. The calculation runs were steady state, with a convergence parameter 1.e-4. The maximum number of outer loop iterations allowing convergence was about 40. The turbulence model adopted was the standard κ-ε model. The advection and turbulence numerical schemes have been set as high resolution.

Boundary conditions applied were specified, constant outlet absolute pressure and inlet mass flow rate. The symmetry planes are free slip walls and the outer wall and bars are non-slipping rough walls with a roughness of 5 and 3 µm respectively.

The adopted calculation mesh consisted of nearly 480000 nodes and 2.32 million tetrahedrons to discretize a segment of the fuel assembly 0.61 m long. Since an almost uniform distribution of element sizes was preferred (except in the spacer grid zone), the distribution of finite volumes seems somewhat coarse, as may be appreciated in Figure 6. However, results using this mesh were good. The following section considers these aspects.

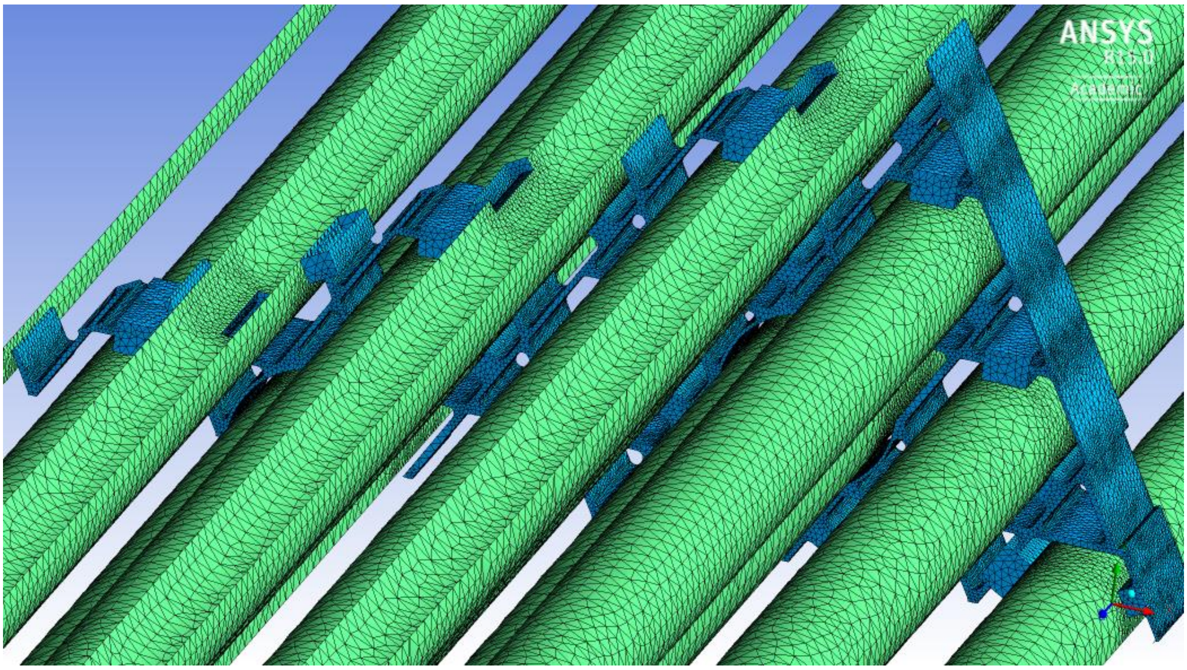

Figure 6 Basic mesh showing surface elements

Considering the above mentioned run settings, a typical run needed 50 minutes to complete in a standard PC.

4. RESULTS AND DISCUSSION

The calculations have been performed for two different components of the fuel element: a) the fuel pin bundle and b) the fuel pin bundle plus the spacer grid. The first simulation allowed the comparison of the calculated pressure drop with some standard data, as shown by Todreas and Kazimi (1989). The average difference in the friction pressure drop was 8% when compared with data for fuel bundles. This agreement permitted to proceed with the calculation of the pressure drop in the spacer grid. As discussed above, the detailed geometry of the springs was not available and a guessed flow restriction was considered, in the form of a fixed separator. This approach and the guessing was correct, as will be shown in what follows. Figure 7 shows this geometry.

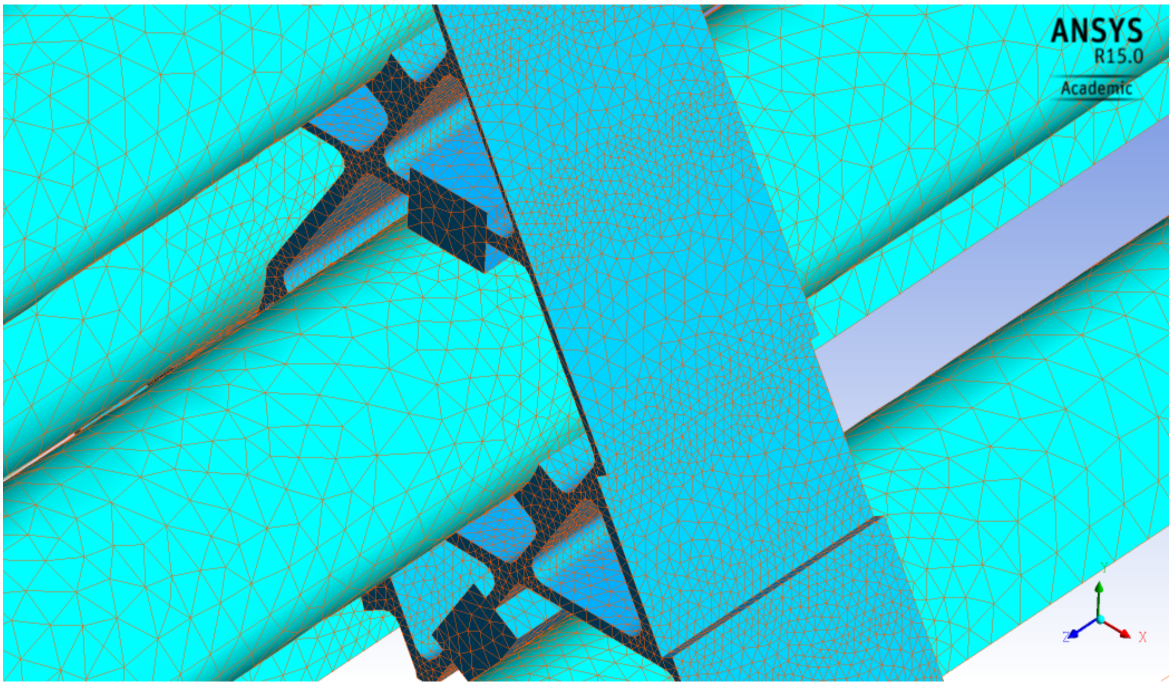

Figure 7 A detail of the adopted, final spacer grid geometry and a sample mesh.

A fuel assembly, similar to the one shown in Figure 1, was tested in the past in a low-pressure experimental facility (1400 kPa) at low temperature (343.16 K). The mass flow rate in experiments ranged from 9 kg/s to 38 kg/s. These thermodynamic conditions are not typical in operation but may allow gaining the searched modelling experience, since flow rates are representative.

Several experiments were conducted in this facility to measure the pressure drop in each fuel channel assembly internal. For instance, some of the tests performed allowed determining the friction pressure drop in the hexagonal channel and in fuel rods, independently. Other experiments were aimed at evaluating the pressure drop in the spacer grids and in the supports. Results concerning the pressure drop in the spacer grid are of particular interest in this approximate study and Figure 8 shows that the simulation performed gave a reasonable result.

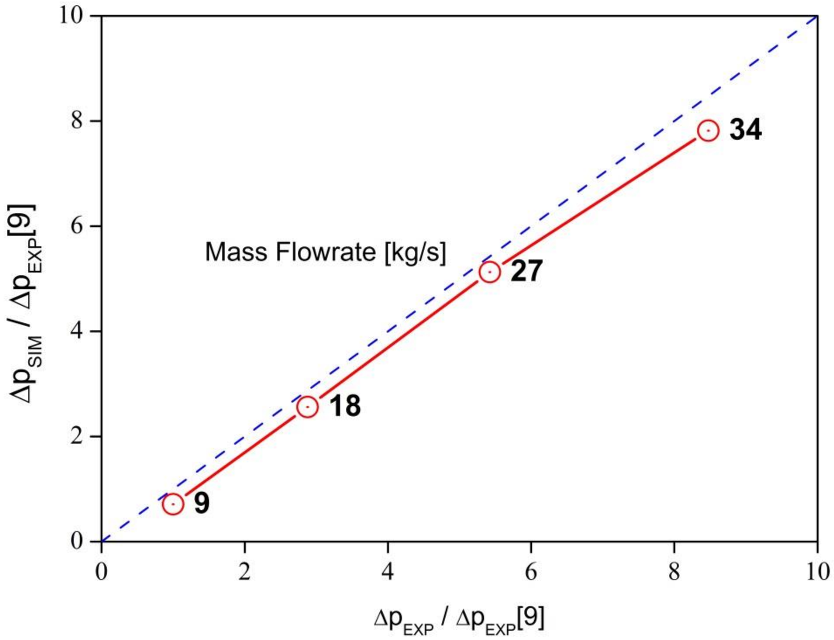

Figure 8 Comparison of calculated and experimental pressure drop in the spacer grid

The simulated experiment results are plotted normalized by the experimental Δp corresponding to a mass flow rate of 9 kg/s. The selected values for comparison correspond to loop mass flow rates (MFR) ≅ {9, 18, 27 and 34} kg/s and shown as labels in Figure 8. The computed pressure losses in the spacer grid differ from the experiment and grow when the MFR diminishes. Corresponding percentages are {-29, -11, -5.5 and -7.8}. These values correspond to the friction factors for each MFR. Using an average friction factor for the MFR range considered, the difference in the concentrated pressure drop coefficient is about 4.5%. The agreement seems good and confirms the premise adopted in the sense that a reasonably fine grid may give good a prediction for a global parameter like pressure drop. Anyway, this agreement may be affected by some error compensation because of the 8% mentioned above in the same range of flow rates for the isolated bars bundle. However, pressure drops are considered per component.

The sensitivity of results to discretization was tested using five grids. Figure 9, drawn for MFR = 27 kg/s, shows that the difference between the simulation results and the experiment seems converged at 5.5% as mentioned above starting from the coarser to the finer grids, which is a satisfactory comparison.

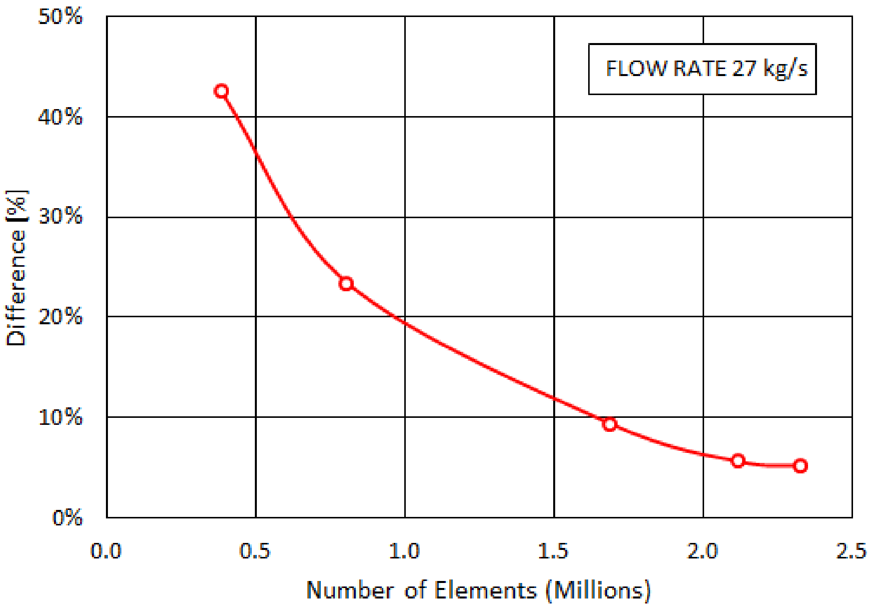

Figure 9 A test of results sensitivity to grid size.

5. CONCLUSIONS

The results presented in this paper are reasonably accurate when compared to experimental data, notwithstanding the approximations to the spacer grid geometry considered and the postulated symmetry of the flow pattern. The overall error was about 8% at the higher flow rates in the experiment. Moreover, the CFD calculations have been performed using standard approximations for turbulence models. The grid convergence tests have shown that the mesh definition was satisfactory for the declared objective of the simulations.

As usual with this type of calculations, the predictions must be guided by standard engineering practices and best practice guidelines and the cases analyzed here were no exception. Most fortunately, the prediction set was included in the experimental domain and this fact allowed searching in a converging way.

A more detailed analysis of the flow pattern in realistic operating conditions, including cross flows and heat transfer will be the subject of future work.

REFERENCES

Please note the following abbreviations:

N14: The 14th International Topical Meeting on Nuclear Reactor Thermal-hydraulics, NURETH-14, Toronto, Ontario, Canada, September 25-30

N15: The 15th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, NURETH-15, Pisa, Italy, May 12-17

CFD4: CFD4NRS-4, Conference on Experimental Validation and Application of CFD and CMFD Codes in Nuclear Reactor Technology, OECD/NEA and IAEA Workshop, 10-12 September, Daejeon, South Korea

ANSYS, ANSYS® Academic Research, Release 15.0, Help System - CFX Guide, ANSYS Inc., 2013

Barthel, F., Franz, R., Krepper E. and Hampel, U., “Experimental studies on sub-cooled boiling in a 3 x 3 rod bundle”, **CFD4**, 2012

Boado Magan, H., Delmastro, D.F. M.Markiewicz, M., Lopasso, E., Diez, F., Giménez, M., Rauschert, A., Halpert, S., M. Chocrón, M., Dezzutti, J.C., Pirani, H., Balbi, C., Fittipaldi, A., Schlamp, M., Murmis, G.M. and Lis, H., “CAREM Project Status”, Science and Technology of Nuclear Installations, Special Issue, “Nuclear Activities in Argentina – 2010”, 2011

Campagnole dos Santos, A.A., Ribeiro, F.L., Costa, F.C., Barros Filho, J.A. and Navarro, M.A., “Verification and validation of a PWR rod bundle segment CFD simulation”, Paper NURETH15-389, **N15**, 2013

Caviezel, D., Narayanan, C. and D. Lakehal, “Highly-resolved LES of turbulent convective flow along a PWR rod bundle”, **N15**, 2013

Chang, S-K., Kim, S. and Song, C-H., "OECD/NEA – KAERI Rod Bundle CFD Benchmark Exercise Test", **CFD4**, 2012

Conner, M.E., Dominguez-Ontiveros, E.E. and Hassan, Y.A., “Hydraulic Benchmark Data for PWR Mixing Vane Grid”, Paper NURETH14-337**, N14**, 2011

Frank, Th., Reiterer, F. and Lifante, C., “Investigation of the PWR subchannel void distribution benchmark (OECD/NRC PSBT BENCHMARK) using ANSYS CFX”, Paper NURETH14-085, **N14**, 2011

Frank, Th., Jain, S., Matyushenko, A.A. and Garbaruk, A.V., "The OECD/NEA MATIS-H BENCHMARK – CFD analysis of water flow through a 5x5 rod bundle with spacer grids using ANSYS FLUENT and ANSYS CFX", **CFD4**, 2012

Karoutas, Z., Yan, J., M. Conner, M. and Mandour, A., “Advanced thermal hydraulic method using 3x3 pin modeling”, Paper NURETH-14 338, **N14**, 2011

Kim, M-H., Tak, N-i. and Noh, J-M., “CFD benchmark study on cross flow experiment in prismatic HTGR core fuel blocks”, Paper NURETH15-674, **N15**, 2013

Krepper, E. and Rzehak, R., “CFD analysis of a void distribution benchmark of the NUPEC PSBT TESTS”, Paper NURETH15-336, **N15**, 2013

Lascar, C., Pierre, M., Goodheart, K., Martin, M., Hatman, A. and Simoneau, J-P., “Validation of a CFD methodology to predict flow fields within rod bundles with spacer grids”, Paper NURETH15-069, **N15**, 2013

Lee, J.R., Kim, J. and Song, C-H, “Synthesis of the OECD/NEA-KAERI rod bundle CFD benchmark exercise”, **CFD4**, 2012

Lencina, L.J. and Ballesteros, H.A., “Thermo-hydraulic behavior of different complex components systems in ATUCHA 2 nuclear power plant using CFD”, Paper NURETH15-548, **N15**, 2013

Melideo, D.F., Moretti, F., Terzuoli, F., D’Auria, F. and Mazzantini, O., “Optimization of the ATUCHA-II fuel assembly spacer grids”, **CFD4**, 2012

Périn, Y., Blyth, T., Avramova, M. and Velkov, K., “Uncertainty analysis of CFD prediction of moderator and fuel parameters for the OECD LWR UAM benchmark, exercise II-3: “bundle thermal-hydraulics”, using the SUSA tool”, Paper NURETH15-426, **N15**, 2013

Petrov, V., Kendrick, B.K., Walter, D. and Manera, A., “Impact of 3D spatial variations in fluid flow on the prediction of crud deposition in a 4x4 PWR sub-assembly”, Paper NURETH15-543, **N15**, 2013

Smith, B.L., Mahaffy, J.H., Bestion, D. and and Zigh, G., “An overview of past and present CFD activities within the framework of WGAMA”, Paper NURETH14-333, **N14**, 2011

Todreas, N.E. and Kazimi, M.S., “***NUCLEAR SYSTEMS I, Thermal Hydraulic Fundamentals***”, Hemisphere Publishing Corporation, 1990

Tóth, S., Kiss, B., Gyuricza, E. and Aszódi, A., “CFD investigation of ALLEGRO fuel assemblies”, Paper NURETH15-672, **N15**, 2013

Yan, J., Conner, M.E., Brewster, R.A., Karoutas, Z.E., Dominguez-Ontiveros, E.E. and Hassan, Y.A., “Validation of CFD method in predicting steady and transient flow field generated by PWR mixing vane grid”, **CFD4**, 2012

Yan, J., Kochunas, B., Hursin, M., Downar, T., Karoutas, T. and Baglietto, E., “Coupled Computational Fluid Dynamics and MOC Neutronic Simulations of Westinghouse PWR Fuel Assemblies with Grid Spacers”, Paper NURETH14-254, **N14**, 2011

Yudov, Y.V., “Using the DINUS code for direct numerical simulation of hydrodynamic processes in VVER-440 fuel rod bundles”, **CFD4**, 2012