

Simulation of PSBT experiment subchannels S1 to S4 using a CFD code and comparison with the experimental results

Deiglys B. Monteiro¹, José R.
Maiorino²

¹deiglysbmonteiro@gmail.com, Departamento de Exatas, Universidade Nove de Julho-
UNINOVE, Av. Professor Luiz Ignácio Anhaia Mello, 1363 - Vila Prudente, São Paulo - SP,
03155-000,

²joserubens.maiorino@ufabc.edu.br,
Programa de pós-graduação em Energia (PPGENE)
Centro de Engenharia e Ciências Sociais Aplicadas (CECS)
Pró-reitoria de Pesquisa (PROPES)
Universidade Federal do ABC
Av. dos Estados, 5001 – Bangú – Santo André, SP, Brazil

1. Introduction

The way in which the coolant flows surrounding a fuel element affects the reactor performance and safety. On the other hand, this flow is affected by the physical properties of the coolant and operational parameters of the reactor such as pressure and temperature. Locally, the temperature of the coolant could rise enough to it boil, reducing the coolant capacity to remove the heat produced in the fuel and giving rise to an accident. The PSBT (PWR Sub-channel and Bundle Test) is an experiment that has been developed to investigate the phenomena occurring in the typical subchannels of a Pressurized Water Reactor (PWR) fuel element submitted to different operational parameters in steady-state or transient regimes, similar to that occurring during transients or abnormal operation in NPPs (Nuclear Power Plants) [1, 2].

The PSBT experiment is main divided in two types: in the former each typical subchannel of a PWR fuel element is tested individually. In this manner, the subchannel does not change mass with the neighbor channels, but only through its inlet and outlet. There are 4 types of subchannels: typical central (S1), typical central with a thimble rod (S2), lateral (S3) and corner (S4), which are schematically represented in Figure 1. The experiment comprises tests in steady-state and transient regimes. On the other hand, in the second type a bundle of 5x5 rods is tested in different conditions, most of them in transient regime. For the bundle, each channel changes mass with the neighbor channels and is affected by mass transfer. This is the most similar condition encountered in a real fuel element, but also a more complex operational condition [1, 2].



Figure 1: Types of subchannels in a typical PWR reactor.
Source: [1]

While the experiments allow to access some important details of the real flow, it involves several practical difficulties related to the instrument accuracy, laboratorial costs, etc. Notwithstanding, the experiments does not allow to access the micro detail of the phenomena such as the differential variation on flow properties such pressure, viscosity, temperature, etc. On the other hand, the CFD (Computational Fluid Dynamics) codes could provide this information as need, not requiring the laboratory structure/apparatus as well as having not the associated costs. By using the CFD codes, the fluid domain is discretized in small volumes to which the governing equations (conservation of mass, momentum and energy) are solved. In addition, the codes could perform simulations and allow to explore different scenarios and operational conditions in a short time and with a high accuracy. Despite of these inherent advantages, the CFD codes uses different kind of models, some of which produce better results in for a geometry and fluid flowing under certain conditions while others not. In this manner, a comparison of the results produced by different models are required, aiming to infer the sensibility of the results when a more complex model is used, which requires a higher computational effort, against the results produced by a simpler model, which on the other hand requires less computational effort [1, 3].

The present work is a development of the work previously conducted by [1] in which an evaluation of the performance of the code THUNDER and of the code ANSYS-CFX[®] CFD was conducted against the experimental data of the PSBT experiment [1]. As reported by the authors, the CFD code had presented good results. However, for the channels S3 and S4, the authors mentioned that additional efforts should be spent aiming to reduce the difference between the simulation results and the experimental data. For this purpose, different turbulence, heat transfer and friction force models could be used. In this manner, this part of the research aims to investigate which turbulence model presents better results than the turbulence model initially used. The main characteristics of the models considered for this evaluation are presented in the section 2.

2. Methodology

The turbulence model used in the simulations performed by [1] was the Shear Stress Transport turbulence model, also known as SST model. According to the authors, this model was setup initially since it is a hybrid model between the κ - ϵ and κ - ω models and thus, could be suitable for different kinds of flows, being also suitable for a first approach when the flow behavior, for a specific geometry/problem/boundary conditions, is unknown [1]. This model is implemented in the ANSYS-CFX[®], which uses a wall function that is automatically adjusted by the code as need. In general, the code performs the follow routine to adjust this function: for regions where there are larger gradients of velocity, pressure, temperature, etc, the model adjusts the function so the model behaviors as the κ - ω model. On the other hand, for the regions where the gradients are smaller, the code adjusts the wall function so the model behaviors as the κ - ϵ model. The way in which the wall function is adjusted is not detailed in the code manual. In a simple manner, the SST acts like the κ - ω model next to walls, edges and curves while it acts like the κ - ϵ in the middle stream (far from wall, edges and curves) [1, 3].

Despite of the advantages of the automatic adjust of the model for a wide variety of situations once it requires less intervention of the user and reduce the time to adjust the wall function properly, some geometries and flows could be not solved properly by the SST model using this automatic adjust depending on the mesh used or boundary conditions. In this manner, a more adequate approach for geometries in which the boundary layer should be solved as detailed as possible, it to adopt the κ - ω model. On the other hand, for geometries in which the boundary layers have small dimensions compared with the dimensions of the fluid domain the κ - ϵ model produces good results [3, 4].

In case of the geometries of the channels S3 and S4, the subchannels have dimensions small enough so the boundary layers fill in all the channel volume. In this manner, and considering the problems mentioned regarding the SST model, a better approach is to simulate these two subchannels using the κ -

ω model instead the SST model, which could also require more refined meshes [3, 4].

In the view of the issues mentioned, new runs for these two subchannels would be done, initially keeping the original mesh, as reported by [1], and refining it as need. The other models and adjusts reported in [1] would be keep unchanged initially and would be adjusted as required, especially the power of the equation that describes the Grace bubble drag model, which had demonstrated some influence over the results. The equation that describes the Grace drag model is given, according the ANSYS-CFX user guide, by equation (1) [1, 3].

$$C_D = r_c^P C_{D\infty} \quad (1)$$

in which $C_{D\infty}$ is the bubble drag coefficient, r_c is the volume fraction of the continuous phase of the flow and P is the exponent value for power law correction [3].

The experimental results considered for the present work development are reported by [1, 2] and is in form of the void fraction (the percentage that represents the volume occupied by gaseous phase with reference to the total volume) and coolant density. Since the objective is to evaluate the agreement of results by using other turbulence models for the subchannels S3 and S4, only the results reported by [1, 2] regarding these two geometries would be considered.

3. Results and Discussion

By changing the turbulence model from the SST model to κ - ω model, it is expected that the flow pattern and boundary layer could be better described than the initial results, even considering that these initial results were considered as good [1]. For reference, the initial simulations had produced the void distributions as presented in Figure 1 for the subchannel S4, the type which the errors reported are larger – in some simulations the errors presented was as larger as 9,2% for void fraction and 13% for density [1]. Despite it is expected that this pattern does not change significantly, any change in the way in which the bubbles formation process and release to the coolant stream as they are drag occurs could lead to a more accurate result, improving the quality of simulations and numerical results and reducing the errors initially obtained.

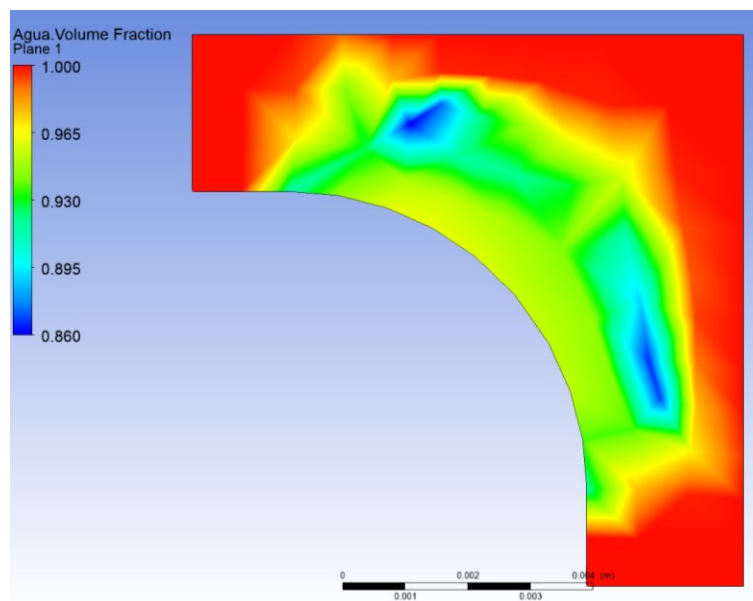


Figure 1: Water volume fraction for the subchannel type S4.

4. Conclusions

Despite the CFD code ANSYS-CFX® has been reported as able to reproduce well the phenomena in the coolant flow for the PSBT experiment, including for the subchannels type S3 and S4, it is possible to that, by changing the turbulence model to another in which the boundary layer could be better described, a higher accuracy result could be obtained. For reference, the results of [1] are being used as benchmark as well as the experimental data of the PSBT experiment [2]. For future works, besides the evaluation of the turbulence model for these two channels, the other channels would be evaluated too. In addition, an evaluation of the influence of the power value of the Grace model over the void fraction and fluid density is planned.

Acknowledgements

The authors are thankful to UFABC for the opportunity to conduct this research as a development of the post-doctoral research conducted previously at the UFABC.

References

- [1] GONZALEZ, D.A.C.; MAIORINO, J.R.; MONTEIRO, D.B.; MOREIRA, J.M.L.; “Thermal-hydraulic validation of two-phase models in THUNDER code against benchmark results and CFD codes”, *Nuclear Engineering and Design*, vol. 369, 110827 (2020).
- [2] OECD-NEA/NRC, *Based on NUPEC PWR Sub-channel and Bundle Test (PSBT): Volume 1 – Experimental database and Final problem specifications*. Report, OECD-NEA, 2012.
- [3] ANSYS-CFX®, “User Guide CFX® Release 20.1”. ANSYS®, 2020.
- [4] SCURO, N. L.; *Numerical simulation of a slow loss of coolant accident in a research nuclear reactor*. Master Thesis, IPEN, Brazil, 2019.