

ANALYTICAL AND EXPERIMENTAL ANALYSIS ON SAFETY RELATED ASPECTS OF THE RMB RESEARCH REACTOR

A. Belchior Jr¹, A. A. C. dos Santos², R. L. Freitas³, H. V. Soares⁴, F. C. Junqueira¹, J. G. Mantecón¹, M. Mattar Neto¹, S. C. Menzel¹, W. M. Torres¹, P. E. Umbehaun¹

¹Instituto de Pesquisas Energéticas e Nucleares (IPEN/CNEN), São Paulo, Brazil

²Centro de Desenvolvimento de Tecnologia Nuclear (CDTN/CNEN), Belo Horizonte, Brazil

³Centro de Coordenação do Reator Multipropósito Brasileiro (RMB/CNEN), São Paulo, Brazil

⁴Amazonia Azul Tecnologias de Defesa S.A. (Amazul), São Paulo, Brazil

E-mail contact of main author: abelchior@ipen.br

Abstract. This paper presents some numerical and experimental safety related activities developed at the Brazilian Multipurpose Reactor (RMB) project by CNEN research institutes. Brief comments on the models and results are presented with emphasis to their relation to the safe design and operation of the reactor. Thermal-hydraulic analysis for Siphon Breaker of the Core Cooling System (CCS); pools hot water layer; core chimney of CCS and spent fuel transport cask are presented, showing results, advantages, difficulties and drawbacks for each analyzed case. All are very distinct cases, involving phenomena that range from two-phase flow and thermal-stratification to lead melting. Beside the one-dimensional thermal hydraulic system Code RELAP5, Computational Fluid Dynamics (CFD) is shown to play an important role in the analysis being performed as it can detail the flow and temperature fields of complex components and phenomena, which are extremely difficult to model analytically or experimentally. Two experimental circuits designed to test RMB fuel elements performance are also presented.

Key Words: RELAP5, CFD, Thermal Stratification, Two-phase flow

1. INTRODUCTION

The Brazilian Multipurpose Reactor – RMB will provide Brazil with a fundamental infrastructure for the national development on activities of the nuclear sector in the areas of social, strategic, industrial applications and scientific and technological development. The Brazilian Nuclear Energy Commission (CNEN) is the reactor owner and operator. The project is in the detail engineering design phase, which is being done by the engineering companies INVAP (from Argentina) and AMAZUL (from Brazil). The RMB is a 30 MW pool type research reactor, that uses MTR type fuel assemblies. It has a square array core with 23 fuel assemblies and two irradiation positions, operating with upward flow and average velocities nearly 10 m/s in the fuel plates channels.

RELAP5/Mod3.3 [1] code has been used for RMB thermal-hydraulic transient and accident calculations. It was reported in the literature that in some cases the use of a CFD code should be necessary since RELAP5 code could not capture their complex 3D phenomena. In [2], for an example, RELAP5 fail to represent the siphon breaker experiments realized at POSTECH facility. The results presented in this paper, on the other hand, showed a very good agreement for both RELAP5 and CFD codes. However, cases involving thermal stratification, as observed in hot water layer system and core cooling system chimney, cannot be easily represented by RELAP5 models without any special trick. For these cases, CFD analysis usually is more suitable.

Other analytical and experimental activities have been developed at the RMB project by CNEN research institutes aiming the verification, validation and qualification of the different structures/systems/components acting in RMB reactor safety.

2. SIPHON BREAK PERFORMANCE STUDIES

The siphon breaker is an important safety feature for a research reactor that, if correctly employed, avoids the drainage of the pool in case of specific pipe breaks in a LOCA. These devices could be a pipe, a valve or an orifice that allows air ingress in the discharging line, which would break the siphon effect that would drain the pool. The main issue with this approach is that there is an undershot height for the water level, meaning that even when air

enters the circuit, the siphon isn't immediately broken, leading to a level of drainage beyond the desired. Knowledge of this undershooting height is crucial for LOCA analysis and design of the siphon breaker.

The POSTECH Siphon Breaker Test Facility [2] consists of an upper tank, a main drainage pipe, discharge valves and pipes, a lower tank and a siphon breaker line or orifice. A schematic diagram of the test facility is shown in FIG. 1-a.

2.1. Validation of RELAP5 mod3.3 Code for Siphon Breaker Simulation

The FIG. 1-b shows a comparison of the results presented in [2] against the RELAP5/Mod3.3 code model developed in [3], called RELAP5/RMB. No special modeling trick needed to be applied to the RELAP5 nodalization. The only care was about the size of the control volume used in the nodalization. The one-dimensional RELAP5/RMB results showed a perfect agreement against the POSTECH experimental result, even much better than the 3-D CFD CUPID code results.

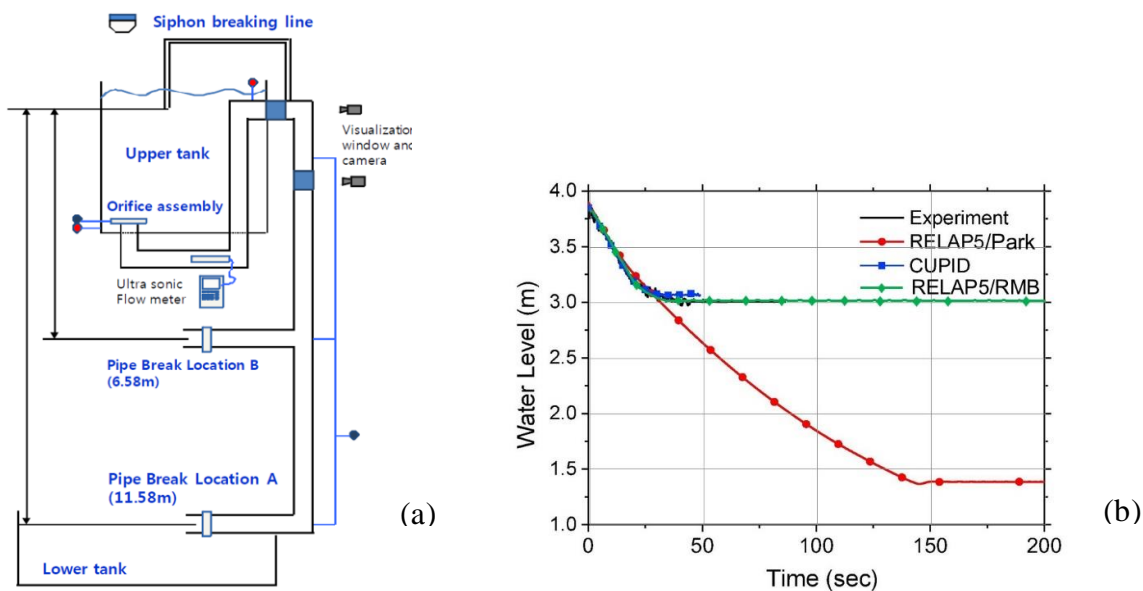


FIG. 1. (a) POSTECH Siphon Breaker Facility; (b) RELAP5/RMB against results shown in [2].

2.2. ANSYS CFX Analysis of Siphon Breaker Performance

To study this phenomenon, a CFD model was defined and validated against experimental data. ANSYS CFX code was used with a two-phase mixture model to obtain results that matched, within uncertainty bounds, the experiments. FIG. 2 demonstrates the obtained CFD results and experimental comparison. The full verification and validations of the numerical procedure as details of the model can be seen in [4].

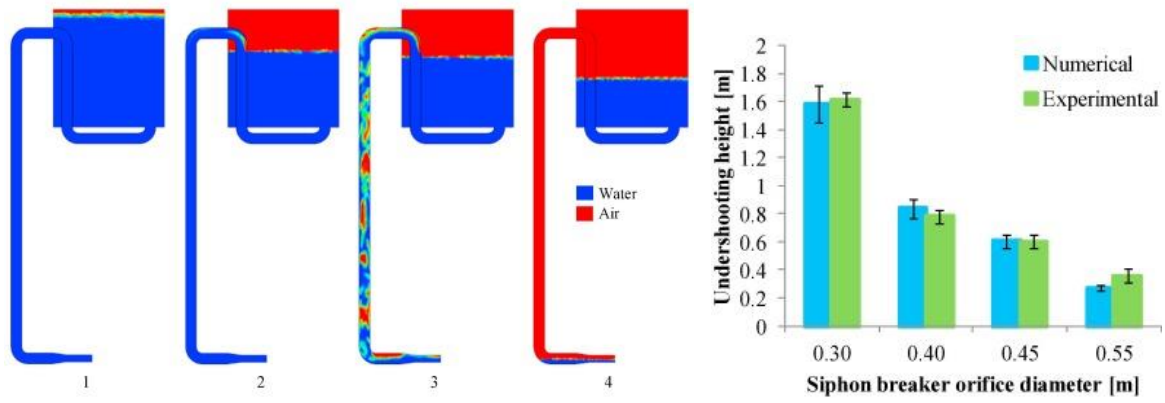


FIG. 2. Siphon breaker dynamics (Left) and undershooting heights for different siphon breaker diameters between simulation and experiment (right) [2].

2.3. RELAP5 Simulation RMB Siphon Breakers after a LOCA in the CCS

A RELAP5/Mod3 code model was developed for thermo-hydraulic simulation of the RMB to analyze the phenomenology of the Siphon Breakers device (four flap valves in the cold leg and one open tube for the atmosphere in the hot leg) during a Loss of Coolant Accident (LOCA) at different points in the primary circuit. Different simulations were carried out at different location in the Core Cooling System (CCS) of the RMB, for example: LOCA before the CCS pumps with and without pump trip and LOCA after the CCS pumps and the heat exchanger. In all RELAP5/Mod3 code simulations, the Siphon Breaker device's performance after a LOCA was effective to allow enough air to enter the outlet pipe of the CCS in order to break the siphon effect and preventing the pool level from reaching the riser (chimney) and the RMB core discovering. In all cases, the reactor pool level stabilized at about 5.5 m after the end of the LOCA simulation, as presented in FIG. 3. The fuel elements were kept underwater and cooled. The simulation details are presented in [5].

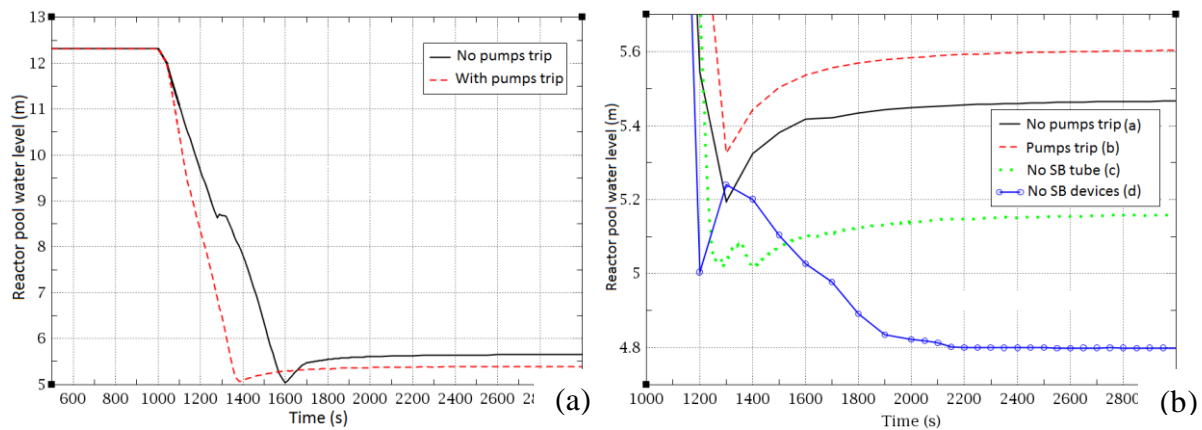


FIG. 3. Reactor pool water level for ruptures at pump suction line^(a) and at heat exchanger outlet^(b.).

3. TWO PHASE FLOW AND THERMALLY STRATIFIED CFD ANALYSIS FOR RMB DESIGN

Thermally stratified and two-phase flow problems are reoccurring in nuclear reactor design. For the RMB project this is not different. Over the past decade CFD codes and models have matured and can be very useful for analysis of these types of flows in complex geometries allowing detailed analysis of local flow characteristics as global design parameter evaluation. In this section we describe Three cases that these complex flows were a challenge during design

stage of the RMB reactor and required the use of CFD simulations to model and allow assessment of the design.

3.1. Hot Water Layer (HWL) system

The HWL system protects the workers and visitors of the reactor facility from receiving radiation doses while the reactor is operational creating a thermally stratified layer of purified hot water that prevents activated elements reaching the surface of the reactor pool. This water layer acts as a radiation shield and the attached purification system guarantees that there will be no buildup of activated elements near the pool surface.

The stability of the HWL can be threatened by certain operational transients, such as the reactor startup process. During these transients the HWL could be partially or completely destroyed, leading to an increase of radiation dose in the reactor room that can last for many hours depending on the systems efficiency to recover. Due to this characteristic, the knowledge of the dynamics of the HWL is crucial for the design of the system and defining operational procedures.

For the RMB, a CFD simulation procedure using ANSYS CFX code and a scaled down experimental model were defined for the study of the HWL. Results obtained numerically [6] and experimentally [7] enabled the analysis of the proposed design under different operational conditions which lead to the definition of a more efficient HWL system.

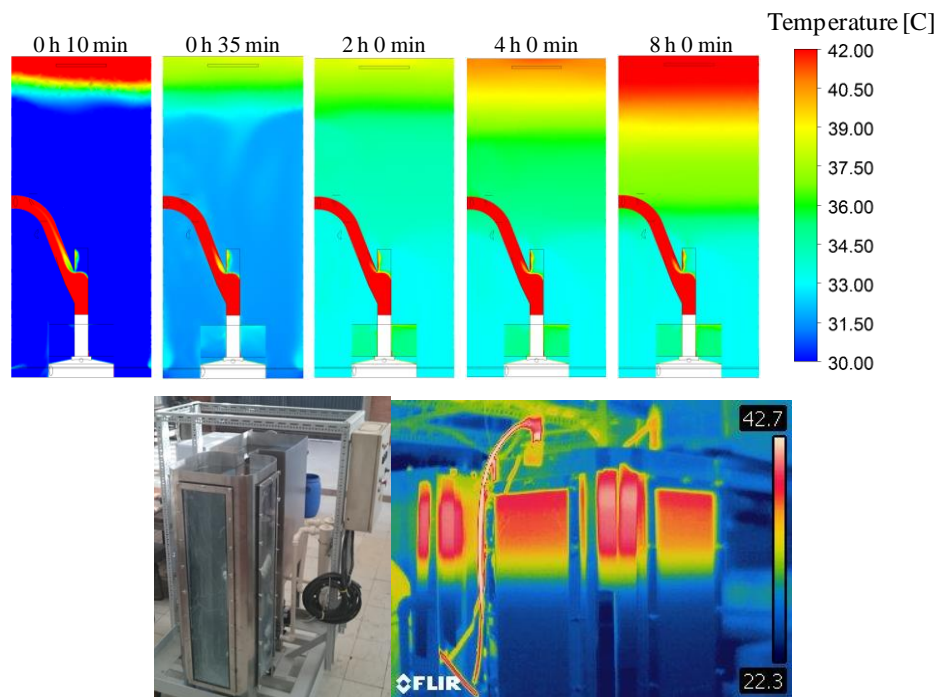


Fig. 4. HWL Numerical simulation (top) [6]; Scaled down facility experimental results (bottom) [7].

3.2. Core Cooling System Chimney

The chimney of the RMB provides access to the core from the top of the pool and provides the necessary passage of flow for natural circulation cooling while the reactor is at shutdown condition. The downside of the chimney is that it connects directly the primary cooling system to the pool. In case of mixture of the primary flow in the pool, the radiation level of the pool would spike due to highly activated elements could crossing the maximum neutron flux area. To avoid this, a percentage of the flow that is removed by the primary cooling system is pulled

from the pool and an interconnection system to the pool cooling system returns this flow to the pool. To study the flow characteristics and dynamics of this thermally stratified problem, an ANSYS CFX CFD model was developed. A mesh analysis was performed and different turbulence models were evaluated [8]. Figure 3 shows results obtained with two turbulence models; it can be seen that very distinct results were obtained. The results so far show that there could be contamination of the pool in function of some operating conditions, however due to the variability of the numerical results, experiments would be necessary to validate which is more accurate for this problem.

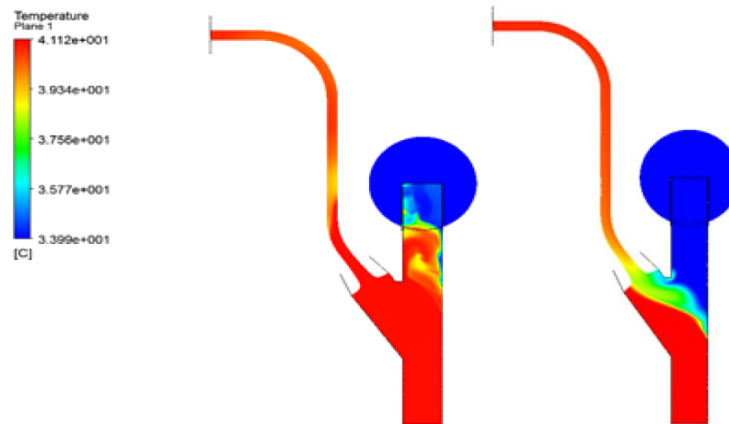


FIG. 4. Simulation of chimney dynamics using SST (left) and k - ϵ s (right) turbulence models [8].

3.3. Dual Purpose Cask (DPC) for fresh and spent fuel elements

A dual-purpose cask for transportation and storage of research reactor fresh and spent fuel is being developed for the RMB. This type B package has to withstand mechanical and thermal design tests. For the thermal test the cask is subjected to an 800°C fire during 30 minutes and subsequent cool down at room temperature. At the end of the test the cask is analyzed for leaks and loss of shielding. To assist in the design of the cask an ANSYS Fluent CFD model was developed. The model included melting modeling due to the presence of lead on the cask structure. Results show that the convection heat transfer of the melted lead in the shielding structure is crucial for the analysis of the temperature profile. Details of the model and results obtained can be seen in reference [9]. Primary flow and pool water interactions at the core chimney

4. EXPERIMENTAL PROGRAM ON THE HYDRODYNAMICAL BEHAVIOR OF FUEL ASSEMBLIES

Test experimental facilities are important to study the phenomenological analysis of the behavior of the RMB fuel assemblies under normal operating conditions, and can evaluate their performance through resistance tests and under extreme conditions, such as critical flow velocity and the influence of flow induced vibration on the fuel plates. For this extreme condition, which must be avoided during the operation of the reactor, plastic deformations of the fuel plates can occur, reducing the flow area and the cooling conditions of the channel, which can cause overheating and damage to the plates with release of radioactive material.

Two experimental circuits are being designed: COLIBRI, for fluid structure analysis in fuel assemblies flow channel and ORQUIDEA for entire MTR fuel element test.

4.1. COLIBRI Experimental Circuit

Based on the circuit developed in [10], the COLIBRI experimental circuit, shown in FIG. 5-a, has been designed to make tests allowing the studies of the fluid-structure phenomenology of flat fuel plates similar to those of the RMB fuel assemblies when subjected to high flow velocities which can induce fuel plates deformation/vibration due to pressure differences in the adjacent flow channels on each side of a fuel plate. The COLIBRI has as main objective to provide experimental data of the fields of speed, pressure and temperature between the fuel assemblies of RMB in order to validate the numerical procedures related to CFD (Computational Fluid Dynamics) codes under an accidental condition.

The FIG. 5-b shows a preliminary structural response studies of the fuel plate's behavior, [11] and [12], performed using a Finite Element Analysis model generated by ANSYS Mechanical. The pressure loadings caused by the fluid flow were calculated using a Computational Fluid Dynamics model created with ANSYS CFX. The fluid-structure interactions will be verified for different channel configurations. In this circuit, vibrations and collapse of the dummy fuel plates will be tested. Experimental data will be compared with CFD (Computational Fluid Dynamics) calculations.

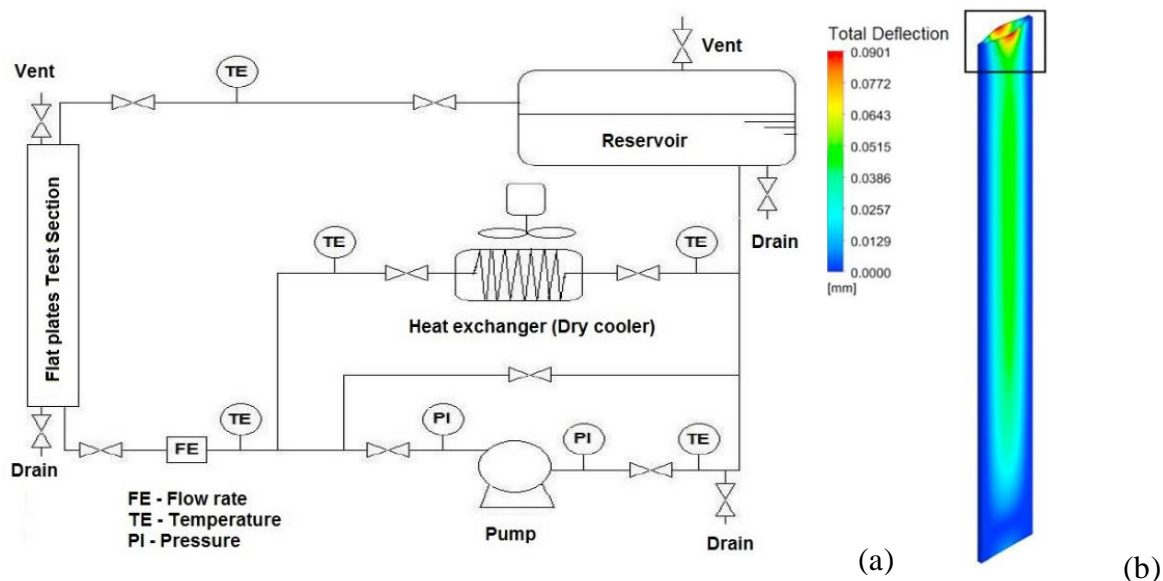


FIG. 5. (a) Schematic drawing of the COLIBRI experimental circuit [13]; (b) Fuel plate deflection calculated by ANSYS fluid-structure interaction [11].

4.2. ORQUÍDEA Experimental Circuit

The ORQUÍDEA experimental circuit [13], presented in FIG. 6, is being designed in order to verify and provide data and information about the hydraulic behavior of "Material Testing Reactor (MTR)" fuel assemblies (FAs) of the RMB and IEA-R1 fuel assemblies in nominal and abnormal conditions at low pressure and temperature. This information will be very important for the licensing process of the fuel assembly before its use in the reactor core. This circuit permits upward and downward flow through the fuel assembly. Its maximum flow rate and pressure are 260 m³/h and 12 MPa(g), respectively. The dynamical behavior of the fuel assemblies and its components will be tested. The main circuit variable to be measured are flow rate, temperature, pressure and differential pressure. Endurance and critical flow velocity tests will be performed. Dummy fuel assemblies will be instrumented with pressure, strain-gages

and flow velocity instruments. The obtained results can be used to validate CFD fuel assembly models [14].

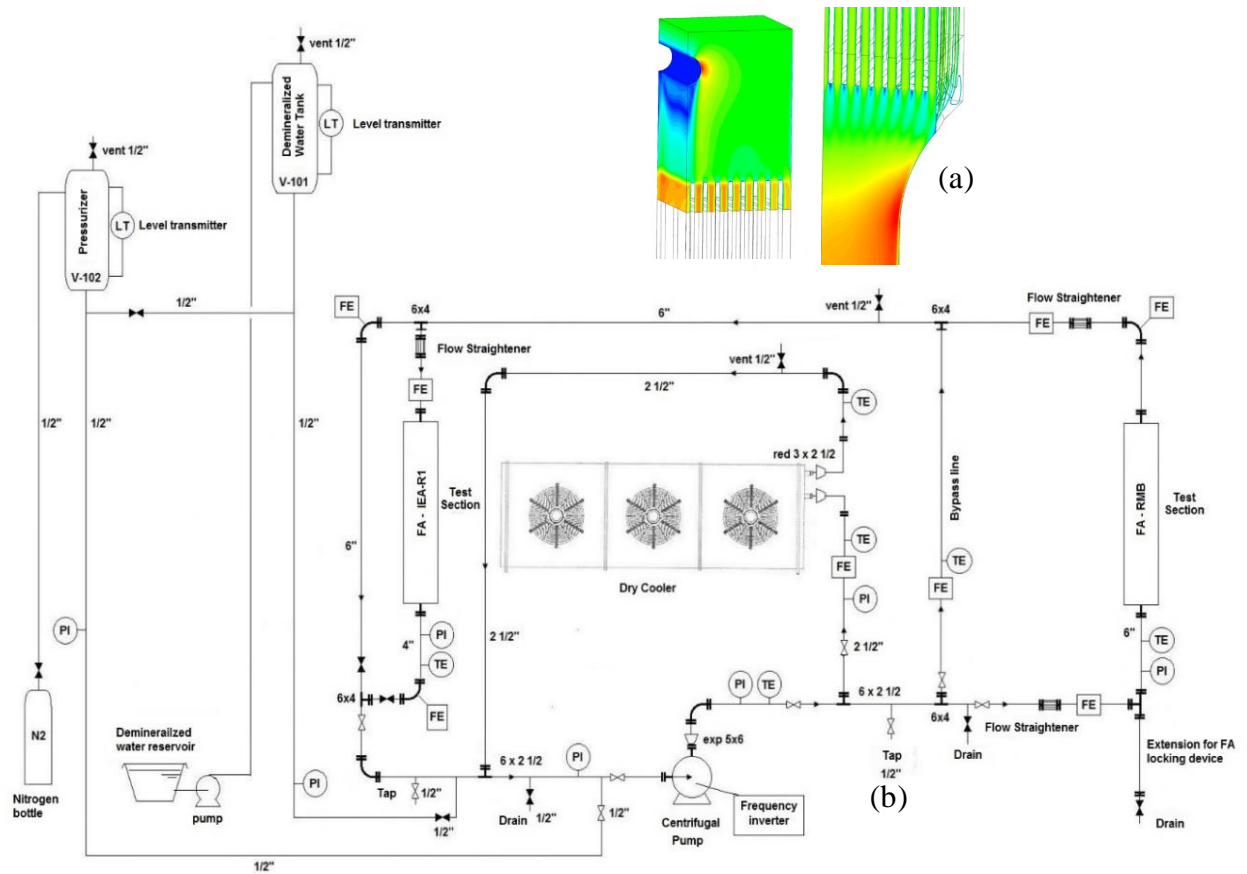


FIG. 6. (a) Velocity profiles at inlet and outlet of a typical MTR fuel element [14];
 (b) Schematic drawing of the ORQUÍDEA experimental circuit [13].

5. CONCLUSION

Some safety related activities developed by CNEN research institutes for RMB project were presented. RELAP5 and ANSYS-CFX codes capabilities to model siphon breaker devices were verified against POSTECH experimental results and the correct actuation of RMB siphon breakers were demonstrated by RELAP5 calculation. Some important RMB components were analyzed through 3D CFD codes. The conceptual design of two experimental circuits to study hydrodynamical were also shown.

REFERENCES

- [1] US NRC, United States Nuclear Regulatory Commission, “RELAP5/MOD3.3 Code Manuals,” Idaho National Engineering Laboratory, 2001.
- [2] PARK, IK KYU; YOON, HAN YOUNG; BEOMPARK, HONG, “Numerical approach to siphon break phenomena in a research reactor pool using the CUPID code,” Nuclear Engineering and Design, vol. 326, pp. 133-142, January 2018.
- [3] BELCHIOR JR, A.; SOARES H. V. ; FREITAS R. L., “Validation of the Relap5 Code for the Simulation of the Siphon Break Effect in Pool Type Research Reactors,” em International Nuclear Atlantic Conference - INAC 2019, Santos, SP, Brazil, 2019.
- [4] MATOZINHOS, C. F.; SANTOS, A. A. C., “Two-phase CFD Simulation of Research Reactor Siphon Breakers: A Verification, Validation and Applicability Study,” Nuclear Engineering and Design, vol. 326, pp. p. 7-16, 2018.
- [5] SOARES, H. V.; BELCHIOR JR, A.; FREITAS, R. L., “RELAP5 Modeling of a Siphon Break Effect on the Brazilian Multipurpose Reactor,” em International Nuclear Atlantic Conference - INAC 2019, Santos, SP, Brazil, 2019.
- [6] SCHWEIZER, F. L. A.; SANTOS, A. A. C. ; PEREIRA, C. ; COSTA, A. L. ; COSTA, A. C. L. ; VELOSO, M. A. F., “CFD Aided Analysis of a Scaled Down Model of the RMB pool,” em International Nuclear Atlantic Conference - INAC 2013, Recife, PE, Brazil, 2013.
- [7] TOMAZ, G. C. Q.; SCHWEIZER, F. L. A. ; CARVALHO, H. M. ; SILVA, V. V. A. ; COSTA, A. C. L. ; SANTOS, A. A. C., “Experimental investigation in the Brazilian multipurpose reactor pool,” em International Nuclear Atlantic Conference - INAC 2015, São Paulo, SP, Brazil, 2015.
- [8] PEREIRA, J. V. M. R., SANTOS, A. A. C., SILVA, V.V.A., CAMPOLINA, D.A.M., BARROS, G.P., “Análise do Escoamento de Água do Sistema de Refrigeração do Reator Multipropósito Brasileiro,” em International Nuclear Atlantic Conference - INAC 2019, Santos, SP, Brazil, 2019.
- [9] RIBEIRO, N.L., VIEIRA, T.A.S., SANTOS, A.A.C., SILVA, V.V.A., CAMPOLINA, D.A.M., “Estudo da Influência da Convecção do Chumbo Fundido na Análise Numérica de um Ensaio Térmico de um Casco de Duplo Propósito,” em International Nuclear Atlantic Conference - INAC 2019, Santos, SP, Brazil, 2019.
- [10] ALVIN DE CASTRO, A. J., Experimental Analysis of Critical Velocity in Flat Plate Fuel Element for Nuclear Research Reactors, São Paulo: (Ph. D. Thesis) Instituto de Pesquisas Energéticas e Nucleares, 2017.
- [11] MANTECÓN, J. G.; MATTAR NETO, M., “Numerical methodology for fluid-structure interaction analysis of nuclear fuel plates under axial flow conditions,” Nuclear Engineering and Design, pp. 76-86, 2018.
- [12] J. G. MANTECÓN e M. MATTAR NETO, “Numerical Analysis on Stability of Nuclear Fuel Plates with Inlet,” Nuclear engineering and Design, pp. 240-248, 2019.
- [13] TORRES, W. M.; UMBEHAUM, P. E.; MATTAR NETO, M. ;BELCHIOR JR, A.; FREITAS, R. L., “RMB Experimental Program on the Hydrodynamical Behavior of Fuel Assemblies,” em International Nuclear Atlantic Conference - INAC 2019, Santos, SP, Brazil, 2019.
- [14] P. H. G. SANTOS, Numerical modeling of a fuel element for hydrodynamic studies, São Paulo: (Master Dissertation) Instituto de Pesquisas Energéticas e Nucleares, 2018.