Nuclear Engineering and Design 326 (2018) 7–16

Contents lists available at ScienceDirect

Nuclear Engineering and Design

journal homepage: www.elsevier.com/locate/nucengdes

Two-phase CFD simulation of research reactor siphon breakers: A verification, validation and applicability study

Camila Freitas Matozinhos[1], Andre Augustos Campagnole dos Santos[*,1]

Comissão Nacional de Energia Nuclear – Centro de Desenvolvimento da Tecnologia Nuclear – (CNEN/CDTN), Av. Presidente Antônio Carlos, 6.627, 31270-901 Belo Horizonte, Minas Gerais, Brazil

ARTICLE INFO

Keywords:
CFD
V & V
Siphon breaker
RMB
Research reactor

ABSTRACT

During a nuclear reactors design phase many possible accidents are postulated. The main accident that could occur in an open pool reactor, such as the Brazilian Multipurpose Reactor (RMB) is the Loss-Of-Coolant Accident (LOCA). This type of accident occurs when the primary cooling piping breaks and the pressure gradient induces the refrigerant (water) to flow through this pipe by siphon effect, draining the pool. This accident can be avoided with a siphon breaking mechanism, which could consist of an orifice in the coolant pipe at a level above the minimum required water level in the pool. If the pipe breaks, air will enter through this orifice, interrupting the siphon flow. However, tests have shown that the mechanism in most cases does not stop the water flow immediately, causing a reduced water level. The siphon break phenomenon is a complex two-phase transient flow. Experimental studies of the siphon break are very expensive and time-consuming tasks. In this case, it would be appropriate to make a Computational Fluid Dynamics (CFD) analysis of this phenomenon, allowing a detailed evaluation of the flow characteristics in a simpler and cheaper way than an experiment. However, the complexity of this phenomenon imposes difficulties to its computer modeling and its accuracy must be assessed. In this study, a methodology is proposed for the simulation of orifice type siphon breaker for research reactors. In order to properly evaluate the numerical methodology, an experimental model available in the literature was simulated and a Verification and Validation (V & V) procedure was performed based on the Grid Convergence Index (*GCI*) method. Numerical uncertainties due to mesh were evaluated and estimated for other simulations with different siphon breaker orifice diameters to assess the applicability of the proposed methodology. Results showed good agreement with experimental results with reasonable uncertainty range. The proposed simple numerical uncertainty estimation procedure aims to encourage the discussion towards an uncertainty quantification culture in the CFD community.

1. Introduction

Brazil is designing a new research reactor called Reator Multipropósito Brasileiro (Brazilian Multipurpose Reactor) – RMB. It is a multipurpose open pool type reactor with a maximum thermal power of 30 MW (Perrota & Obadia, 2011). The RMB will provide an important infrastructure for national development of the nuclear sector, having a high impact in social, strategic, industrial, medical, scientific and technological development areas (Perrota, 2014).

The design, construction and operation of nuclear reactors follow the highest quality standards and have high reliability conditions. During a nuclear reactor design phase many possible accidents are postulated and their causes, consequences, prevention and mitigation are evaluated. One of these accidents is the Loss-of-Coolant Accident (LOCA), which can be considered the most severe accident in pool type research reactors (Hamidouche et al., 2005).

A LOCA occurs in an open pool research reactor when the primary cooling pipe breaks and the pressure gradient makes the coolant run through this pipe by siphon effect, draining the pool. It is important to ensure that if the pipe breaks, a sufficient water level will remain in the pool to prevent a major accident. This can be done by a siphon breaking mechanism, which usually consists in making an orifice or connecting a pipe in the primary circuit refrigeration line at a pool water level to guarantee sufficient reactor cooling. If the cooling line breaks, air will enter through the siphon breaker interrupting the flow. However, observations have shown that the level of remaining water is lower than the siphon breaker level (Seo et al., 2012). The length between the siphon breaker orifice level and the final water level in the pool after the

* Corresponding authors.
E-mail addresses: camilafreitasmatozinhos@gmail.com (C.F. Matozinhos), aacs@cdtn.br (A.A. Campagnole dos Santos).
[1] Authors contributed equally.

http://dx.doi.org/10.1016/j.nucengdes.2017.10.022
Received 30 January 2017; Received in revised form 17 October 2017; Accepted 25 October 2017
0029-5493/

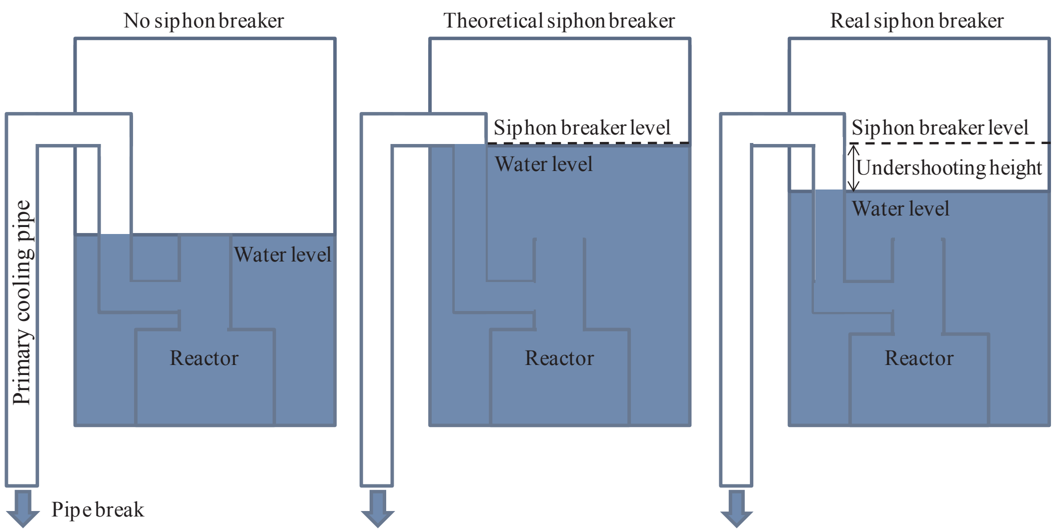

Fig. 1. Post LOCA scenarios.

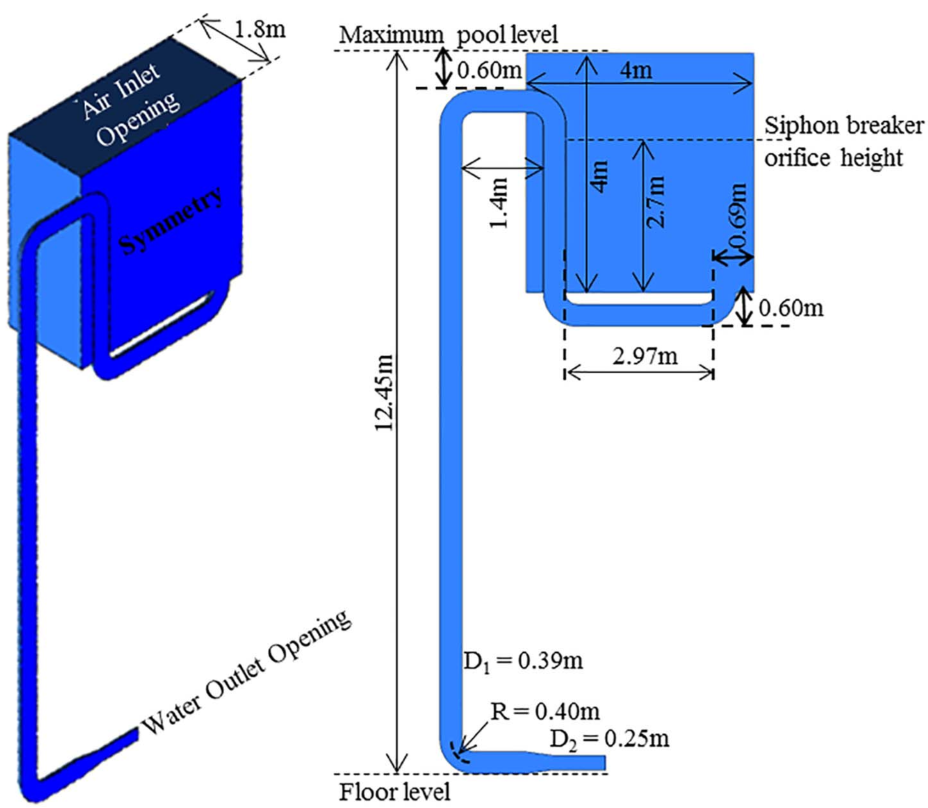

Fig. 2. Geometry (right) and boundary conditions (left) used in the simulations.

siphon breaks is called undershooting height. This is an important issue since the inventory of remaining water is critical for post-accident evaluations.

Fig. 1 illustrates the consequences of a LOCA in an open pool research reactor for three conditions: when there is no siphon breaker and the pool is drained, in the theoretical scenario when the siphon breaker stops the water flow in the exact moment that air starts to enter the pipe and in the real scenario when an undershooting height is observed even with the siphon breaker.

The siphon break phenomenon is a complex two-phase transient flow and is influenced by various parameters such as mixing, drag, turbulence and pressure losses. Experimental studies of the siphon break are very expensive and time-consuming tasks as is show in the work of Kang et al. (2013).

An alternative to an experiment are theoretical and numerical analysis. In the work of Lee & Kim (2016) a theoretical model is formulated to predict the undershoot level of water post a LOCA in open pool research reactors. For more detailed analysis in complex geometries a Computational Fluid Dynamics (CFD) methodology is an alternative. Seo et al. (2012) has shown the potential of using CFD methodology for the study of the siphon breaker. They have modeled a pipe type siphon breaker having obtained satisfactory results compared to experiment. Further CFD studies are still required to evaluate mesh and modeling parameters, numerical uncertainties and other siphon breaker geometries.

The objective of the study is to asses a CFD methodology for the analysis of orifice type siphon breakers applying a Verification and Validation (V & V) (Roache, 2009) procedure to evaluate numerical