Annals of Nuclear Energy 114 (2018) 311–317

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Numerical simulation of drainage performance in a drain device of steam generator

Yong Mei [a], Shengjie Gong [a,*], Chi Wang [a], Hanyang Gu [a], Bingbin Ying [b], Yingxi Song [b]

[a] School of Nuclear Science and Engineering, Shanghai Jiao Tong University, Shanghai, China
[b] Shanghai Nuclear Engineering Research and Design Institute, Shanghai, China

ARTICLE INFO

Article history:
Received 23 April 2017
Received in revised form 11 December 2017
Accepted 12 December 2017

Keywords:
Drainage performance
Steam generator
Free surface
Turbulence model
Two phase flow

ABSTRACT

As the last stage of a moisture separator, the drain device is an important part of the secondary separator which can affect the separation efficiency. The emphasis of this paper is placed on studying the flow structure and predicting the free liquid surface height in the drain device. Four different turbulence models including $k-\varepsilon$, $k-\omega$, Shear Stress Transport (SST) and Scale-Adaptive Simulation (SAS-SST) were used to simulate the drainage process with air-water mixture (cold state) under ambient temperature and atmosphere pressure. These models were validated by the experimental data obtained in our previous study. Simulation results showed that the free liquid surface height predicted by the SAS-SST model is in best agreement with the experimental results under cold state. Therefore it was further used to perform the simulation for steam-water mixture (hot state) under the standard working condition (138.1 m^3/h, 7 MPa and 285.8 °C). The maximum free liquid surface height is simulated to be 269.3 mm for the hot state by the SAS-SST model which is 4.5% higher than that of the cold state. However, it could also meet the design object.

1. Introduction

Moisture separator is a vital device in the steam generator for pressurized water reactors. It provides high quality dry steam for turbine and protects the turbine from the impact of droplets. A generally accepted industry standard quotes that the mass flow rate of water droplets in the steam should be less than 0.1% in order to minimize damage to turbines and keep a relatively higher plant efficiency (Green and Hetsroni, 1995).

A typical moisture separator consists of three parts, including primary separator, gravity separator and secondary separator. The primary separator is usually a swirl vane separator and can remove more than 80% of the water liquid. Above the primary separator, there is a space called gravity separator which separates most droplets and makes the steam to flow uniformly. Usually, the water droplets in the steam should be less than 8%–10% before entering into the secondary separator. The separation process of the secondary separator is done by a chevron dryer including chevron vanes and a drain tank. Water droplets separated by the chevron vanes will be collected by a vertical gutter drain pockets and fall into a drain tank located under the lower ends of the chevron vanes. Extensive experimental and theoretical researches have been performed on each main part of the moisture separator. The flow pattern inside the swirl-vane separator and the separation performance were studied by Matsubayashi et al. (2012) and Xiong et al. (2013) through conducting visualization experiments. The gravity separation process was studied by Prabhudharwadkar et al. (2010) and a correlation for predicting the phenomenon of carryover in a separator drum was proposed. The performance of chevron vanes with different structures were experimentally and numerically studied by Nakao et al. (1998, 1999; Li et al. (2007). Considering the whole water droplets separation process, Zhang et al. (2016) simulated the droplet-laden flows in moisture separators based on the Lagrangian–Eulerian approach and studied the physical mechanism of the moisture separator.

Although researches mentioned above have provided much insight into the droplet separation process in the moisture separator, it should be noted that few studies focus on the performance of the drain tank, which is also an important component of the secondary separator. The drain tank is used to collect the separated water by the chevron vanes and drain the water down to attend the recirculation. The height of the drain tank should be big enough to prevent the overflow of separated liquid which will lead to the re-entrainment of the droplet and deteriorate the dry steam. But from the view of economics, minimizing the drain tank height could shorten the steam generator and cost down. In our previous

* Corresponding author.
E-mail address: gsj@sjtu.edu.cn (S. Gong).

https://doi.org/10.1016/j.anucene.2017.12.037
0306-4549/

study, Gu et al. (2016) conducted visualization experiments with air-water mixture under room temperature and atmosphere pressure (cold state) to investigate the flow conditions and drainage performances of drain devices with different structures. However, whether these experimental results are valid under the real operation condition (hot state) still needs to be verified.

Considering the great difficulties and high cost to perform such an experiment under the hot state, the numerical simulation method is proposed. As the key parameter to evaluate the drainage performance, the free liquid surface is determined by two aspects: one is the liquid jets impinging from the perforated plate above the tank, which was studied by Hammoumi et al. (2002), Qu et al. (2011) and Grosshans et al. (2014); the other is the dips formed during water draining through the drain pipe at the bottom of the tank, which was studied by Park and Chang (2011) and Basu et al. (2013). To capture the gas-liquid interface, the VOF (Volume of Fluid) method and ALE (Arbitrary Lagrange Euler) method are the most common choices. Agarwal et al. (2014) used the VOF method and Psihogios et al. (2015) used the ALE method to simulate the draining process in a tank with a drainage port in the bottom, respectively. Brouilliot and Lubin (2013) chose the VOF method to simulate the free surface in the tank for a liquid jet plunging process.

In this study, the free surface model which is similar to the VOF method was applied and four different turbulence models (k-ε, k-ω, SST, SAS-SST) were tested under the cold state which could be validated by the experimental data obtained in our previous study (Gu et al., 2016). Then, the validated model would be chosen to simulate the drainage performance in the drain device under the hot state. The flow characteristics such as gas entrainment and void fraction distribution in the drain device were analyzed and the factors which affect the drainage performance of the drain tank were discussed.

2. Description of numerical model

2.1. Governing equations

The cold state simulation is performed by the CFX with the version of ANSYS 16.0. In the simulation, each phase is regarded as a continuous fluid and the homogeneous model is used. A compres-

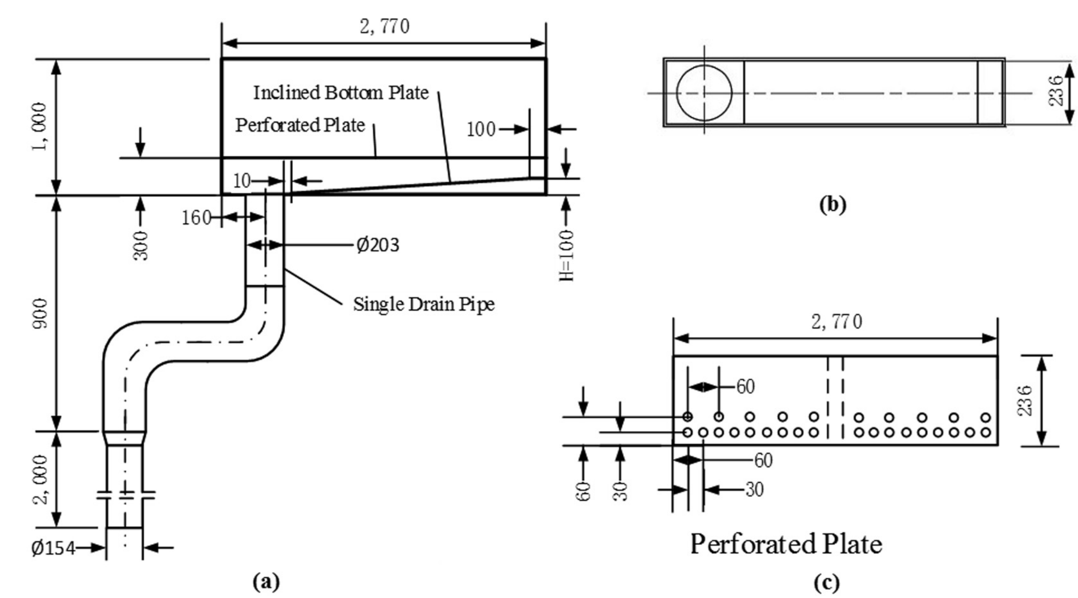

Fig. 1. The schematic of test section (a. Front view of the test section; b. Top view of the test section; c. Distribution of water entrance holes on the perforated plate).

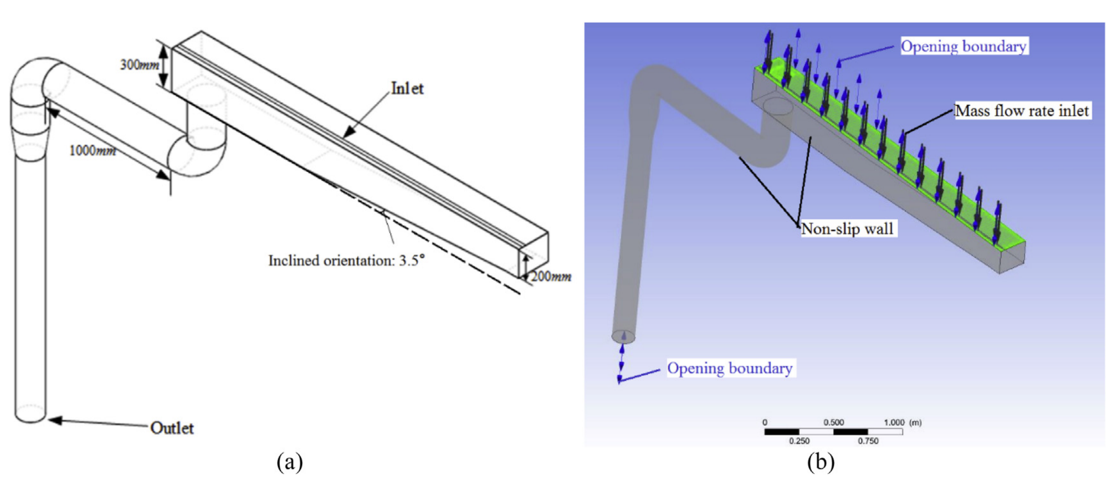

Fig. 2. The computational domain (a) and boundary conditions (b).