Annals of Nuclear Energy 92 (2016) 136–149

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Evaluation of thermal hydraulic safety of a nuclear fuel assembly in a mast assembly of nuclear power plant

YoungSoo Kim [a,1], HuiYung Kim [a], JinYeong Bak [a], JaeJun Jeong [a], JaeYeong Jeon [b], ByongJo Yun [a,*]

[a] School of Mechanical Engineering, Pusan National University, Keumjeong-gu, Busan 609-735, South Korea

[b] Power MnC Co., Ltd., Habuk-myeon, Yangsan-si, Gyeongsangnam-do 626-862, South Korea

ARTICLE INFO

Article history:
Received 18 August 2015
Received in revised form 30 November 2015
Accepted 20 January 2016
Available online 10 February 2016

Keywords:
Spent fuel bundle
Computational Fluid Dynamics
Natural convection analysis
Critical Heat Flux (CHF)
Departure from the nucleate boiling ratio (DNBR)

ABSTRACT

We investigated the thermal–hydraulic safety of spent fuel bundles installed in a mast assembly, assuming a lock-up accident of a fuel transfer system. For this, flow analysis with Computational Fluid Dynamics (CFD) for the natural convection and subsequent safety evaluation with 1-D Nuclear Power Plant (NPP) safety analysis code were carried out. Prior to the natural convection analysis for the mast assembly using CFD code, we performed benchmark calculations against two experimental data sets obtained by Betts and Bokhari (2000) and PNL (Pacific Northwest Laboratory, 1980). This was done to select the proper physical models for a natural convection flow analysis and to ensure the reliability of prediction of natural convection flow in the fuel bundle geometry. Finally, we performed a main natural convection analysis for the fuel assembly inside the mast assembly. From this calculation, we observed a stable natural circulation flow between the mast assembly and pool side, and obtained coolant velocity at the inlet of the spent fuel bundle. This flow condition is given as a boundary condition for the 1-D NPP safety analysis code, which is used to predict the Critical Heat Flux and then departure from the nucleate boiling ratio.

1. Introduction

In typical Korean Nuclear Power Plants (NPPs), a third of the nuclear fuel is replaced roughly every 18 months, depending on the burning of the fuel. Refueling is completed by removing the old nuclear fuel and then installing the new fuel in the reactor by a fuel-handling system, after a cold shutdown of the NPP. The fuel replacement system consists of a refueling machine, a fuel-transfer system, a fuel elevator installed in the containment, a new fuel elevator, and a spent-fuel handling system in an independent fuel storage building. The mast assembly is composed of a hoist box and mast (see Fig. 1) and is the major equipment used to protect fuel assemblies from damage and to transfer them during the refueling process. In this study, we studied the thermal hydraulic safety of fuel bundles installed in the mast assembly.

Under nominal conditions, the mast assembly containing the spent fuel bundle is moved in a pool in which decay heat is transferred from the spent fuel to the coolant by natural convection. If the amount of decay heat removed is insufficient, the spent fuel can melt down as the fuel rods of the core did in the Fukushima Daiichi accident. The thermal–hydraulic safety of the mast assembly therefore should be evaluated under the conditions existing in a postulated failure of the fuel transfer system.

Safety evaluation and flow analyses for the nuclear spent fuel bundle in the pool were performed by previous investigators with one dimensional (1-D) best estimated safety analysis codes and Computational Fluid Dynamics (CFD) codes.

Kaliatka et al. (2010) used a best-estimate system thermal hydraulic code, RELAP5, to evaluate the evaporation of coolant, uncovering, and heat-up processes of spent fuel assemblies, in a scenario assuming a 'loss of heat removal' accident in the spent fuel pool of the Lithuania Ignalina NPP (Nuclear Power Plant). Wang et al. (2012) also performed a thermal hydraulics analysis of the spent fuel pool for Chinshan NPP (China) by employing both the system analysis code TRACE and CFD code. In et al. (2013), performed benchmark calculations of Multichannel Analyzer for steady states and Transients in Rod Arrays (MATRA), MARS (Multi-dimensional Analysis of Reactor Safety), and CFX-10 codes against the 8-by-8 fuel bundle tests by Nuclear Power Engineering Corporation (NUPEC) of Japan. They reported that MATRA and MARS showed reasonable capability for calculation of the void fraction in the subchannel. In contrast, CFX-10 showed lower void fraction prediction results than found in the experimental

* Corresponding author. Tel.: +82 51 510 2484; fax: +82 51 514 7460.
E-mail address: bjyun@pusan.ac.kr (B. Yun).

[1] Current address: Korea Institute of Nuclear Safety, Department of Reactor System Evaluation, 62 Gwahak-ro, Yuseong-gu, Daejeon 305-338, South Korea.

http://dx.doi.org/10.1016/j.anucene.2016.01.033
0306-4549/

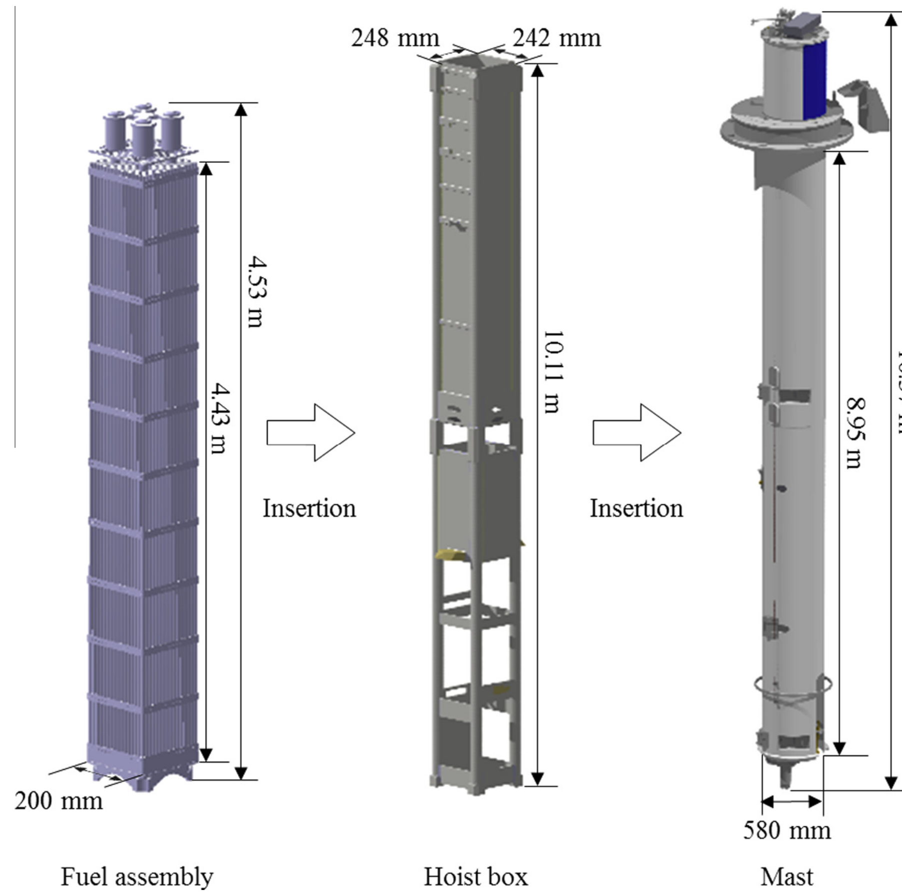

Fig. 1. Three-dimensional view of mast assembly for fuel transfer system.

data. M. E. Conner et al. (2012) validated their single-phase CFD methodology with STAR-CD in the 5-by-5 subchannel which simulates a PWR fuel bundle under operating conditions. Moon et al. (2005) benchmarked MARS code for the prediction of Critical Heat Flux (CHF) and reported that the maximum uncertainty of the predicted CHF with Look-Up Table (LUT) method is 25% in the 3×3 bundle under the low mass flux condition. In addition to these, efforts were also made to apply CFD code to multi-dimensional two-phase flow analyses for the fuel bundle. Tentner et al. (2005, 2007) proposed advanced two-phase flow models using CFD code, for the analysis of the Boiling Water Reactor (BWR) fuel assembly. Lo and Osman (2012) applied 3-D CFD methodology to predict the boiling two-phase flow in one of the PSBT 5×5 rod bundle tests.

For safety issues regarding spent fuel in the storage system, requirements (Ryu, 2009) are suggested by KINS (Korea Institute of Nuclear Safety), the regulatory body in South Korea. The requirements are as follows: (1) Spent fuels must be cooled in water under all conditions. (2) The top of the fuel must be located at least 3 m below from the surface in the case of failure of the inlet, outlet, or piping equipment. (3) The coolant must be maintained at temperature of less than 60 °C.

In this study, we tried to evaluate the safety of spent nuclear fuel inside a MAST assembly for Korean Advanced Power Reactor 1400 (APR1400) in a postulated accident. For this, we assumed that the mast assembly stopped in the pool for a given reason (e.g., mechanical failure or loss of electricity) during transfer of the spent fuel, and then appraised the safety of the spent fuel. To evaluate the thermal–hydraulic safety under the conditions of the assumed accident, we first tried to investigate whether the natural convection flow was sufficient to transfer decay heat from the spent fuel to the coolant. This was done using CFD code and then by quantifying the thermal margin by calculation of the CHF using the best-estimate NPP safety-analysis code. The overall strategy for these evaluations is summarized in Fig. 2. As shown in the figure, the STAR-CCM+ Ver. 9.02., a commercial CFD code was at first benchmarked against two sets of experimental data. This was done for selection of the proper physical models, development of an optimized calculation mesh, and validation of the prediction capability for the subchannel under natural convection flow. Then, this was applied to the natural convection flow analysis of the spent fuel in the pool. Through the analysis, we confirmed stable natural convection flow in the pool and then calculated the inlet velocity of coolant in the accident condition. Subsequently, the CHF was predicted by the (Jeong et al., 1999) code developed by KAERI (Korea Atomic Energy Research Institute) using the inlet velocity of coolant as a boundary condition. Finally, the thermal–hydraulic safety was quantified by calculating the departure from the nucleate boiling ratio (DNBR) under these flow conditions.

2. Benchmark calculations of CFD code for natural convection flow

In the present study, the commercially available CFD code, STAR-CCM+, was chosen for analysis of the natural convection flow of the mast assembly located in the containment pool. In the CFD analysis, applying a proper turbulent model is requisite for successful flow analysis. In the conventional CFD codes, a turbulence model such as Detached Eddy Simulation, k–ε, k–ω, or the Reynolds Stress Model based on Reynolds-averaged Navier–Stokes