Nuclear Engineering and Design 295 (2015) 374–387

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Full-scale 3-D finite element modeling of a two-loop pressurized water reactor for heat transfer, thermal–mechanical cyclic stress analysis, and environmental fatigue life estimation☆

Subhasish Mohanty*, William K. Soppet, Saurindranath Majumdar, Krishnamurti Natesan

Argonne National Laboratory, Lemont, IL, USA

HIGHLIGHTS

- Full-scale 3-D finite element model.
- Pressurized water reactor.
- Heat transfer analysis.
- Thermal–mechanical stress analysis.
- Environmental fatigue life estimation.

ARTICLE INFO

Article history:
Received 18 May 2015
Received in revised form 2 October 2015
Accepted 4 October 2015

ABSTRACT

This paper discusses a system-level finite element model of a two-loop pressurized water reactor (PWR). Based on this model, system-level heat transfer analysis and subsequent sequentially coupled thermal–mechanical stress analysis were performed for typical thermal–mechanical fatigue cycles. The in-air fatigue lives of example components, such as the hot and cold legs, were estimated on the basis of stress analysis results, ASME in-air fatigue life estimation criteria, and fatigue design curves. Furthermore, environmental correction factors and associated PWR environment fatigue lives for the hot and cold legs were estimated by using estimated stress and strain histories and the approach described in US-NRC report: NUREG-6909.

1. Introduction

System-level computer modeling of complex nuclear systems is increasingly becoming a trend due to the availability of advanced multi-physics computer programs and the increasing use of multiprocessor-based parallel computing hardware and software. Recently, many works have been published on thermal–hydraulics simulations of fluid flow and heat transfer in a single reactor component or in a complex large-scale assembly (DOE; Palmtag et al., 2014; Kang et al., 2011; Conner et al., 2013; Yoon et al., 2012; Murase et al., 2010; Shan et al., 2014). This type of system-level thermal–hydraulics model helps to better understand and to accurately predict the fluid flow and heat transfer not only in individual components but also the overall system and the interaction with each other. Along a similar line, computational structural mechanics analysis is increasingly being used to perform stress and fracture mechanics analysis under complex component/assembly-level multi-axial stress states. For example, recent advances in 3-D finite element analyses (FEA) code and associated improvements in multi-physics modeling capability (e.g., thermal–mechanical stress analysis) and fracture mechanics simulation capability allow more accurate 3-D stress and structural integrity analysis of reactor components not only under combined thermal–mechanical loading but also under multi-axial component/assembly-level stress states (Lin et al., 2006; Yu et al., 2002; Jia et al., 2014). In addition to the above-mentioned multi-physics thermal–mechanical stress analysis, the

☆ The submitted manuscript has been created by UChicago Argonne, LLC, Operator of Argonne National Laboratory ("Argonne"). Argonne, a U.S. Department of Energy Office of Science laboratory, is operated under Contract No. DE-AC02-06CH11357. The U.S. Government retains for itself, and others acting on its behalf, a paid-up nonexclusive, irrevocable worldwide license in said article to reproduce, prepare derivative works, distribute copies to the public, and perform publicly and display publicly, by or on behalf of the Government.

* Corresponding author. Tel.: +1 630 252 5137.
E-mail address: smohanty@anl.gov (S. Mohanty).

http://dx.doi.org/10.1016/j.nucengdes.2015.10.012
0029-5493/

present-generation FEA code also allows determination of the effect of other field variables, such as the effect of neutron dose on the 3D stress state of reactor structural components (Mohanty et al., 2012, 2013a). Furthermore, advances in FEA tools for 3-D fracture mechanics and crack propagation allow accurate prediction of the structural integrity of reactor components under severe accident conditions, such as loss-of-coolant accidents (LOCAs). For example, propagation of preexisting stress corrosion cracking (SCC) in steam generator tubes and the associated rupture pressure can be predicted accurately under LOCA (Mohanty et al., 2013b). Similarly, in reactor pressure vessels and other primary pressure boundary components, the effect of pressurized thermal shock (PTS) under severe accident conditions can be predicted by using 3-D FEA tools (Chen et al., 2014a; Qian and Niffenegger, 2013a,b, 2015; Keim et al., 2001; González-Albuixech et al., 2014). Both LOCA and PTS conditions are key elements in the integrity evaluation of nuclear reactor components and require a multidisciplinary effort to link the thermal–hydraulic analysis results to structural and fracture mechanics models. In addition to the multi-physics capability, the current-generation FEA code also allows one to model complex time-dependent material effects. For example, time-dependent creep damage of the reactor pressure vessel under severe accident conditions, such as a LOCA, can be more accurately predicted by using component-scale, 3-D FEA models (Villanueva et al., 2012).

The above-mentioned structural analysis examples based on FEA are mostly restricted to a single component under static or quasi-static transient loading. However, a few studies have been done involving thermal–mechanical fatigue modeling using system-level 3-D models. Also, at present, most of the work related to fatigue evaluation in reactor environments is based on stress analysis at the individual component level combined with estimation of the associated fatigue life using stress/strain life curves (Chopra and Stevens, 2014; Japan Nuclear Energy Safety Organization, 2011; Gray and Verlinich, 2012; Chen et al., 2014b). However, as part of the Light Water Reactor Sustainability (LWRS) program sponsored by the Department of Energy (DOE), Argonne National Laboratory (ANL) is trying to develop a more mechanistic-based fatigue evaluation approach (Mohanty et al., 2013c, 2014) under realistic multi-physics and multi-axial stress states. Under this program ANL is trying to develop an assembly level finite element (FE) model for system-level stress analysis and associated fatigue life evaluation under thermal–mechanical cyclic loading. For the purpose, in the present work, we developed preliminary FE models for a Westinghouse-type two-loop pressurized water reactor (PWR). Based on the FE models, system-level thermal–mechanical fatigue (TMF) analyses were performed. Furthermore, these TMF results were used for in-air and environmental fatigue life estimation of some example components such as the reactor cold and hot legs. The related model and calculated results are discussed below.

2. Finite element modeling

2.1. System level 3-D solid model

Finite element models were developed for system-level TMF analysis of a typical two-loop PWR. The models were developed by using commercially available ABAQUS FE software (Dassault Systèmes, 2014). The FE models were based on approximate geometry determined from publicly available literatures (Shah and MacDonald, 1993; Schulz, 2006; Cummins et al., 2003; Westinghouse Electric, 2000, 2011). In the assembly-level model, only major reactor parts such as the pressure vessel, steam generator outer shell, and hot and cold leg pipes were considered. Fig. 1 shows the resulting assembly-level 3-D solid model of the 2-loop PWR considered in this work.

For simplicity, the surge line and pressurizer were not considered in the assembly-level model. Also, a simplified coolant pump model was assumed, and only the top section that connects both the steam generator and cold leg was considered. However, in the future, for more detailed analysis, the surge line, pressurizer and other important components will be considered. The assembly-level model was developed by using 3-D solid models of individual components with single or multiple sections. The 3-D models were developed by using ABAQUS CAE software. The individual sections or components were appropriately constrained to maintain their locations with respect to the global assembly. In the assembly model, the individual sections were tied together by using tie constraints. The bottom section of the reactor pressure vessel (RPV) was tied to a base plate, which was attached to the ground and constrained in all directions. Similarly, the coolant pumps were tied to additional base plates. However, in contrast to the RPV base plates, the coolant pump base plates were only restricted in the vertical direction and were allowed to move along both horizontal directions. These constraints are shown in Fig. 1. This condition was designed to mimic the real reactor conditions, allowing free thermal expansion. However, note that the above boundary conditions are simplified assumptions and do not necessarily represent the exact boundary conditions in a real reactor. For example in a typical Siemens designed PWR, the RPV is supported at its bottom end by an inverted frusto-conical surface concentric with the axis of the vessel and fixed to its bottom (Domer and Michel, 1976). This surface rests on an upright frusto-conical surface which is also concentric with the vessel's axis. Radial thermal displacements of the RPV's bottom section results in diameter changes in the frusto-conical surface fixed to its bottom so that this surface by cam action slides up and down on the bottom frusto-conical surface. With a properly defined angularity between upper and bottom frusto-conical surfaces, compensates the vertical thermal expansion of the vessel which occurs simultaneously with its radial expansion. In another example, for a Westinghouse designed reactor the main coolant flow nozzles (both hot and cold legs) serve as vessel supports in addition to performing their primary function as conduits (Desmarchais, 1971). The support nozzles rest on integral pads which allow free thermal expansion of RPV. However, in the work discussed in this paper, the details of the real reactor supports or boundary conditions were not considered, rather simplified boundary constraints for reactor supports were modeled as shown in Fig. 1. In addition, in the present assembly-level model, we did not consider the plane of symmetries. In the future we intend to add unsymmetrical components such as a surge line and pressurizer, and it may not be possible to implement a symmetric boundary condition in the system-level reactor model. Hence, in the present model, symmetric boundary conditions were not considered for possible future amendment. The same assembly-level 3-D model was used for both heat transfer analysis and subsequent sequential structural analysis.

2.2. Finite element mesh

The individual components in the reactor assembly were FE meshed by using 3-D brick elements. We chose DC3D8, 8-node linear heat transfer elements to mesh the individual components in the assembly-level heat transfer models. The corresponding C3D8, 8-node linear elements were used for the stress analysis models. The assembly has a total of 85,610 DC3D8 elements for heat transfer models or C3D8 elements for structural analysis models. Table 1 shows the number of elements and nodes used for individual components and the respective material type used in the simulation. Fig. 2a shows the full assembly-level FE mesh of the considered 2-loop reactor. This figure also shows typical ID and OD surface elements. Fig. 2b shows the magnified version of Fig. 2a showing