Annals of Nuclear Energy 115 (2018) 423–429

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Research on the non-uniform inflow characteristics of the canned nuclear coolant pump

Yuehui Wang [a], Pengfei Wang [b], Xiaohong Tan [c], Zhongbin Xu [a,*], Xiaodong Ruan [d]

[a] Institute of Process Equipment, College of Energy Engineering, Zhejiang University, Hangzhou 310027, China
[b] School of Engineering, Zhejiang University City College, Hangzhou 310035, China
[c] College of Mechanical and Electrical Engineering, Hangzhou Polytechnic, Hangzhou 311402, China
[d] The State Key Laboratory of Fluid Power and Mechatronic Systems, Zhejiang University, Hangzhou 310007, China

ARTICLE INFO

Article history:
Received 25 June 2017
Received in revised form 31 January 2018
Accepted 2 February 2018
Available online 22 February 2018

Keywords:
Nuclear coolant pump
Non-uniform inflow
Inlet distortion

ABSTRACT

Computational fluid dynamics is used to analyze the velocity field of the inflow and the pressure loading on the blades of the reactor coolant pumps under different conditions. The Reynolds-averaged Navier-Stokes equations with the k-ε turbulence model were solved to simulate both steady and unsteady states of the inflows. The model was validated with experiments on the heads and efficiencies by testing a straight pipe at different flow rates. With this model, the paper demonstrates that, by replacing the straight pipe with a channel head, the inlet flow field of the impeller becomes non-uniform, forming a swirling flow under lateral pressure differences and a local low-energy region near the tube wall. The inlet distortion destroys the circular symmetry of the flow field and reduces the head and efficiency of the pump. The flow field in the impeller is more complicated and chaotic and imposes an unbalanced force on the impeller thus increasing the risk of fatigue failure of the impeller. Because of the significant effect of the non-uniform inflow induced by the channel head on the performance of the pump, understanding the underlying mechanism is of importance in improving the design of the reactor coolant pump.

1. Introduction

As reported, two incidents happened in Sanmen's Nuclear Power Station in Zhejiang, China in 2015. Subsequent investigations found that severe cracks appeared on the surface of the blades during serial tests of the nuclear coolant pump (NCP), specifically, in the junctions between the inlet edges of the blades and shroud. The flow field of the blade inlet is the most chaotic part of the whole flow field of the impeller. Moreover, the plant operators independently had made tests on the pump and steam generator (SG) before running the whole test.However, while the inflow conduit was canceled in the NCP, the head of the SG was connected to the pump by a very short pipe at the entrance. In this advanced passive PWR system, two canned motor pumps were directly attached to the cold side of the SG (Schul and Terry, 2006), that may greatly worsened the flow field at the inlet of the pump and generated the impeller cracks. Hence, the effect of non-uniform inflow on NCPs needs to be investigated. Rotating components, such as rotating blades, are one of the major sources of operational failure (Zemp et al., 2010). In industrial applications, the inlet section features consisting of pipe bends, struts, and guide vanes, create a non-uniform inlet flow field upstream of the impeller (Greitzer et al., 2007). Cumpsty (2004) investigated a multistage axial compressor and showed that the inlet distortion strongly affects the performance of the compressor. Moreover, resonant vibration of the impeller blades was found to lead to high-cycle-fatigue failure (Ariga et al., 1982; Engeda et al., 2003). However, few important investigations were conducted on hydraulic machinery. In studying the axial flow pump with and without suction chamber; Wang and co-workers noted that the pump and chamber should be coupled to reveal the pump's internal flow mechanism (Wang et al., 2008). In a numerical simulation comparing the performance of axial flow pump under both uniform and non-uniform inflows, Shi and co-workers found that non-uniform flow has a clear influence on the external characteristics, radial load and pressure pulsations (Shi et al., 2014). The complex non-uniform inflow may change pump performance of the pump, either in positively or negatively. An experimental study of the influences on mixed-flow pump performance and radial loads in non-uniform inflow situations found the pump head, torque, and axial forces are

* Corresponding author.
E-mail addresses: wangyuehui@zju.edu.cn (Y. Wang), peng@zju.edu.cn (P. Wang), xuzhongbin@zju.edu.cn (Z. Xu), xdruan@zju.edu.cn (X. Ruan).

https://doi.org/10.1016/j.anucene.2018.02.007
0306-4549/

Nomenclature

D	Diameter, m
v	Velocity, m/s
Q	Flow rate, m^3/s
H	Head, m
η	efficiency
t	Time, s
F_A	Axial force, N
F_R	Radial force, N
ρ	Density, kg/m^3
α	Swirl angle, degree
U_a	Axial velocity, m/s
U_θ	Circumferential velocity, m/s
θ_i^+	Circumferential extent of the positive (co-rotating) swirl region
θ_i^-	Circumferential extent of the negative (counter-rotating) swirl region
SS_i	sector swirl, degree
SI	Swirl intensity
SD	Swirl directivity

reduced (Van Esch, 2009). As for the NCP, experiments on the flow characteristics found a connected SG head and pump is harmful to the flow field (Huang et al., 2002) and the pump head and efficiency decreased (Hou et al., 2016, 2014). However, the interaction between the channel head and the pump has not been investigated since no pumps were connected to the channel head in the test loop. Differences in performance and pressure pulsation with different inflows suggest that the differences in both channel head and straight pipe inflows have significant effects on the pump unsteady pressure pulsation (Yun et al., 2017). In view of the foregoing, the inlet flow field of the NCP with and without the SG head was investigated, specifically at the impeller inlet and the channel head outlet. The circumferential velocity induced by the complex geometry upstream in the SG discharge pipe and under NCP blade loading was also analyzed.

2. Flow modelling and simulation procedure

2.1. Numerical methods

To simplify the calculation, a 1/4 model of the SG head and a single NCP was configured. The flow components included 1/4 SG head, short inflow segment, impeller, guide vane, and pumping chamber. The computational fluid dynamics simulations were performed using the commercial ANSYS Fluent software package. Meshing of the fluid domain is shown in Fig. 1.

The model was meshed using tetrahedra and hexahedra elements. Grid independence was verified: when the total number of grid is above 6.3 million, the change in values of the head and efficiency were less than 1%. Grid properties were eventually determined by a compromise between spatial resolution and the computational power required. Each revolution of the impeller was divided into 360 time-steps, i.e., 1 degree per time step, which sufficed to resolve the different inlet distortion patterns and their convection through the pump stage. For the spatial discretization, a second-order scheme was used. The RNG k-epsilon turbulence model was therefore used in the simulations and the magnitude of the dimensionless wall distance y^+ around the blades was lower than 200. For the steady state case, a constant velocity was applied at the inlet of the domain. Data were obtained from performance measurements in the test facility, and the interface used the reference coordinate system model (Luo and Gosman, 1994). For the time-resolved case, inlet velocities were used as inlet boundary conditions. The interface was meshed using the sliding mesh model (Wang and Calabrese, 1997). At the outlet of the domain, for both cases, the average static pressure over the entire outlet area was iteratively adjusted so that the computed mass flow could be matched to the measured mass flows at the corresponding operating point.

2.2. Scale model pump test

Before beginning simulations, the accuracy of the numerical method and model were verified. A numerical calculation of a scaled pump (1:2.5) was compared with the experimental results obtained running the pump under the same conditions. Impeller and guide vane (see Fig. 2) designed by Shenyang Blower Works

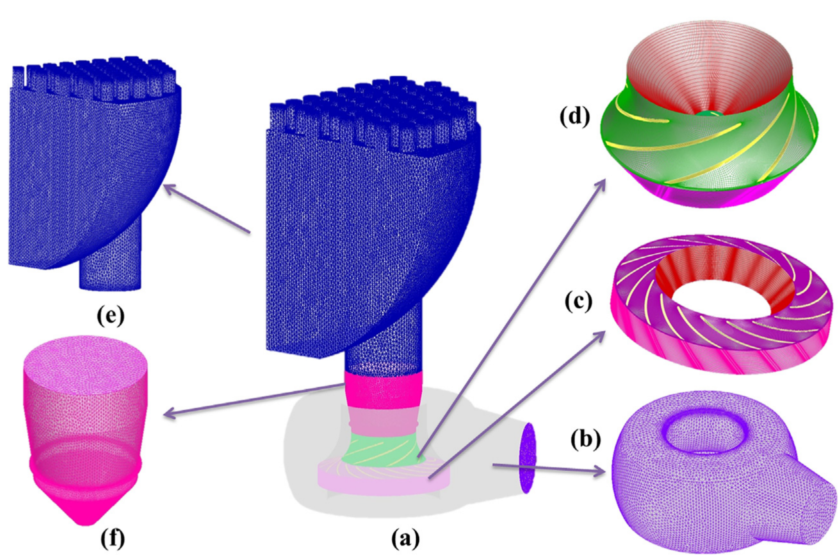

Fig. 1. Grids of the model (a) assembly (b) pump chamber (c) guide vane (d) impeller (e) head of steam generator (f) short xuction pipe.