INTERNATIONAL JOURNAL OF HYDROGEN ENERGY XXX (2017) 1–8

Available online at www.sciencedirect.com

ScienceDirect

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

An optimized CFD method for conceptual flow design of water cooled ceramic blanket

Wenyuan Fan, Changhong Peng, Yun Guo *

School of Nuclear Science and Technology, University of Science and Technology of China, Hefei, 230026, China

ARTICLE INFO

Article history:
Received 28 March 2017
Received in revised form
11 June 2017
Accepted 19 June 2017
Available online xxx

Keywords:
Water cooled ceramic blanket
Conceptual design
CFD
Optimization

ABSTRACT

Computational Fluid Dynamics (CFD) is a significant method for conceptual design of Water Cooled Ceramic Blanket (WCCB), since it could provide three-dimensional results with high accuracy. However, full CFD modeling and simulation toward the whole blanket flow channels is computationally expensive. An optimized CFD method based on partial modeling toward manifold regions is developed to overcome this disadvantage. Several flow conditions inside the First Wall (FW) and Cooling Plates (CPs) are simulated by the proposed method and the full modeling method. The maximum difference in mass flow rate prediction is 1.2%. The highest reductions in mesh size and modeled volume are 64.1% and 75.6% respectively. Results indicate that the newly developed method could provide accurate predictions with little computing resource requirements.

Introduction

The blanket is the bridge for heat transfer and tritium production. The tritium self-consistency is very important in fusion power plant. Tritium production needs suitable temperature in the blanket. Hence, the quick analysis method for flow and temperature fields is necessary. Due to its excellent heat transfer performances, water has been used as coolant for a long time in ranges of fields, including nuclear power plants. Water Cooled Ceramic Blanket (WCCB) is widely investigated [1,2,6,10] and now becoming a potential selection for China Fusion Engineering Test Reactor [7].

Amongst several parts of the conceptual design of WCCB, flow distribution analysis is the primary one and of paramount significance. Because flow design determines the heat removal capacity of the WCCB and subsequently affects the breeding process which is sensitive to temperature. However, a blanket-scale modeling and investigation including flow, heat transfer and breeding process requires huge computing resources. Therefore, it is a good practice to firstly investigate flow phenomenon inside the blanket to provide data for heat transfer and breeding process calculations. Then based on results of such calculations, modifications to the design could be conducted iteratively. During this process, flow, heat and breeding process phenomena could be well understood separately and main factors of each part could be figured out. All these results would contribute to potential larger scale simulations.

System analysis codes, which are initially designed for fission reactors [8], are used to acquire the preliminary understanding of the flow and heat phenomena inside the blanket [11]. However, these codes could only investigate flow distribution one-dimensionally. In order to obtain detailed results, computational fluid dynamics (CFD) method should be employed. CFD method has been widely used in nuclear

* *Corresponding author.*
E-mail address: guoyun79@ustc.edu.cn (Y. Guo).
http://dx.doi.org/10.1016/j.ijhydene.2017.06.150
0360-3199/

system [4,12]. A three-dimensional CFD study has been performed to investigate flow distribution of the side wall of a test blanker module and validated by corresponding experiment by Ref. [9]. In their work, meshes with about 300 million nodes were used, indicating that it would be computationally expensive to conduct CFD simulations even toward a small part of the blanket. Especially, during the conceptual design process, parameters such as dimensions of channels, pitches of channels, shape of manifolds may change frequently according to flow, heat transfer and breeding results based on former design. Then it would take a lot of time using CFD method to totally model and investigate flow phenomenon inside the blanket.

In this paper, an optimized CFD method for flow distribution calculations is developed, which could significantly reduce requirements on computing resources with high accuracy.

Flow scheme configuration

Compared with full CFD modeling, the method proposed in this study could significantly reduce requirements on computing recourses. Even though, results obtained by both methods should be compared to evaluate the newly developed method. However, a full CFD investigation toward all flow channels inside the blanket is computationally expensive. Actually, it is adequate to introduce the developed method by investigating phenomenon inside First Wall (FW) only. Nevertheless, by extending the simulated domain, it could illustrate the application scope of the method. Considering these factors, coolant channels inside FW and Cooling Plates (CPs) are modeled and investigated, as depicted in Fig. 1. Geometry parameters of flow channels are shown in Table 1.

Undeniably, the geometry of coolant channels inside the blanket is really complex. However, when considering the topology of these channels, the flow phenomenon could be simplified, as shown in Fig. 2. Through the inlet of Manifold-1, coolants flow into the blanket with a 5.457 kg/s mass flow rate. Then water flows to Manifold-2 through 42 identical coolant channels inside FW. In Manifold-2, coolants collect and then redistribute to 60 identical flow channels inside CPs. All the coolants finally flow into Manifold-3 and depart from the investigated region through five outlets of Manifold-3.

Table 1 – Geometry parameters of flow channels.

	FW	CP
Number of coolant channels	42	15/CP, 60 in total
Channel cross-sectional dimensions	8 mm × 8 mm	5 mm × 5 mm
Channel pitch	22 mm	15 mm
Channel length (excluding elbows)	~2770 mm	~6305 mm
Gap between CPs	–	11 mm

Methodology

Traditionally, CFD models are constructed and solved toward the whole flow region, as shown in Fig. 1 (a) including all the parts shown in Fig.1 (b), Fig.1 (c) and Fig. 1 (d). This provides detailed CFD results but requires large computing resources at the same time. This method is employed in the present study to evaluate the newly developed method, which is described in the following.

In order to investigate the phenomenon in the proposed way, the flow domain is separated into channel regions (as shown in Fig.1 (b) and Fig.1 (c)) and manifold regions (as shown in Fig. 1 (d)). Manifold regions consist of manifolds and corresponding inlet channels and outlet channels. Each inlet/

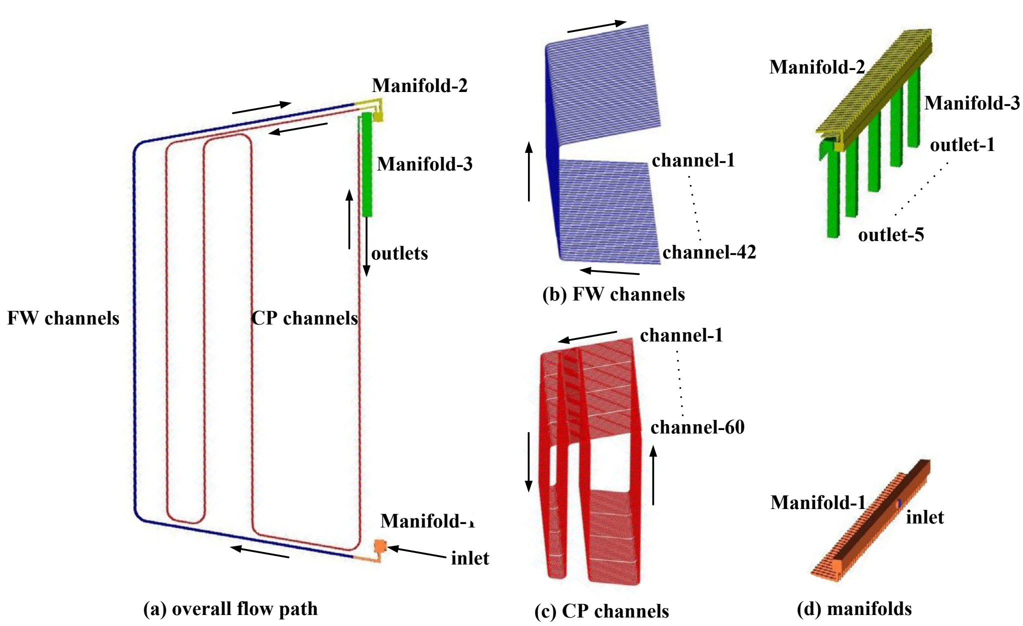

Fig. 1 – Flow channels inside FW and BZ.

Please cite this article in press as: Fan W, et al., An optimized CFD method for conceptual flow design of water cooled ceramic blanket, International Journal of Hydrogen Energy (2017), http://dx.doi.org/10.1016/j.ijhydene.2017.06.150