Fusion Engineering and Design 89 (2014) 94–98

Contents lists available at ScienceDirect

Fusion Engineering and Design

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

Heat transfer study for ITER blanket shield block cooling design

Weishan Kang*, Tao Yuan, Xiujie Zhang

Southwestern Institute of Physics, P.O. Box 432, Chengdu 610064, PR China

HIGHLIGHTS

- Theoretical and numerical methods were developed to quickly evaluate the temperature for the SB design and optimization.
- The methods can be used not only for the cooling design, but also to know about the heat transfer in the SB.
- Comparisons of the results from numerical steady-state thermal analysis with these developed models show that the difference is quite small.

ARTICLE INFO

Article history:
Received 11 June 2013
Received in revised form
22 November 2013
Accepted 10 December 2013
Available online 31 December 2013

Keywords:
ITER
Blanket
Shield block
Heat transfer

ABSTRACT

Two simplified models were developed for the cooling design of ITER shield block. Moreover, a new model, circular cylinder centered in a square solid, was also adopted to estimate the temperature, where the effects of heat transfer coefficient and volumetric heat rate were separated and studied individually. After that, the impact of dimension on the heat transfer in the new model was studied by a series of numerical analyses. At the last part, a numerical steady-state thermal analysis of a typical full shield block (SB) was performed to verify these models. Comparisons of the results from numerical analysis with these models show that the difference is acceptable in the practical application. The methods can be used not only for the cooling design, but also to know about the heat transfer in the SB.

1. Introduction

ITER blanket is one of the key in-vessel components, which play an important role of providing nuclear and thermal shielding to in-vessel, vessel and ex-vessel components. Blanket comprises first wall (FW) panel and shield block (SB), and the main function of SB is to provide nuclear shielding and supply the FW panel with cooling water. The nuclear heat deposited in the SB will be removed by cooling channels. Removing nuclear heat is one of the main drivers for SB design.

Typical SB cooling channels can be seen in Fig. 1. According to the ITER design [1], poloidal cooling was frequently adopted, and it plays the key role to remove the heat. To adapt to the complex interface, radial cooling was also considered. In the central part of the SB, the cooling channels are arranged radially with frontal headers, and these channels connected to inlet and outlet openings are marked as B_{inlet} and B_{outlet}, respectively. In the lateral parts, cooling channels are divided into six sub branches pipes in parallel, marked as B_0–B_5 from center to lateral end of SB, and the coolant is distributed between poloidal channels with headers located on the top/bottom of the SB surfaces. Toroidal drillings connect these sub branches with the coolant inlet and outlet openings. The cooling channels had been arranged mainly in a symmetric way to minimize the thermal distortions and stresses in the SB body.

One of the most challenges for cooling design is the complex interface. SB is required to accommodate FW panel, attachments and the components located on the vacuum vessel, such as in-vessel coils, the diagnostics, coolant manifolds, etc. Moreover, slits are also cut on the front side, from the top and bottom of SB to reduce electromagnetic loads. For this reason, the cut-out due to interface become very complicated. Cooling channels should be designed carefully to make sure that minimum wall thickness is compatible with ITER Vacuum Handbook [2].

Another challenge is to reduce the hot spots and serious thermal gradients of the SB during operation. Hot spot and high thermal gradient will lead to serious thermal stress in the SB, and consequently it will potentially damage the SB [3], which should be avoided.

In this paper, theoretical and numerical methods were developed for the cooling design. At the last part, steady-state numerical thermal analysis of a full SB has been performed. Comparisons of the results from numerical analysis with these developed models were also done.

* Corresponding author. Tel.: +86 28 8285 0419.
E-mail address: kangweishan@swip.ac.cn (W. Kang).

0920-3796/$ – see front matter © 2013 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.fusengdes.2013.12.013

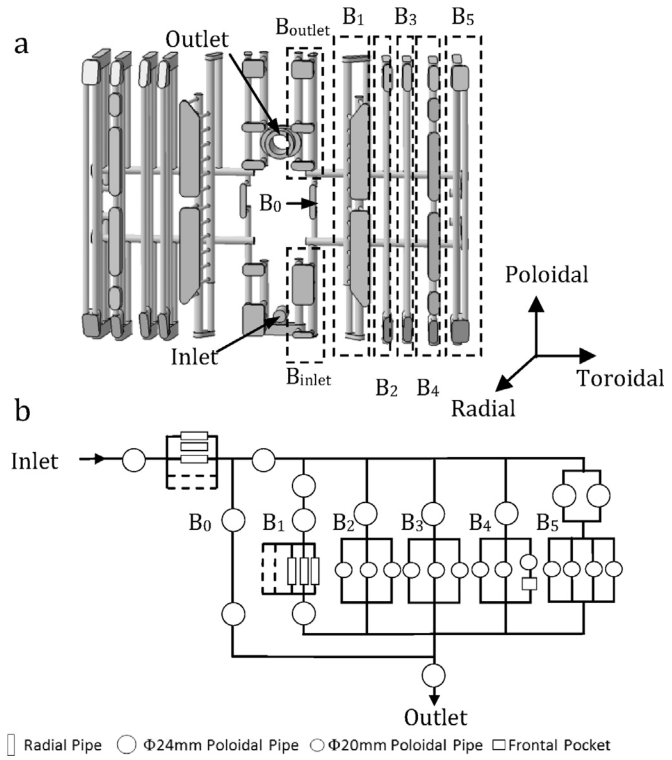

Fig. 1. (a) Cooling system of typical SB; (b) its flow scheme through a right half cut horizontally.

2. Two simplified thermal models

2.1. Model 1: convective heat transfer in the flat plate

Heat transfer in the flat plate is illustrated as Fig. 2, where the heat is generated at a uniform volumetric heat rate Q in the plate, and dissipated by convection from the boundary at $x = d$. Other boundaries are assumed to be isothermal. The fluid temperature is T_1, and the surface temperature at the other side of plate is T_2. The main equation governing the steady-state heat transfer of the plate is:

$$\frac{d^2t}{dx^2} + \frac{Q}{\lambda} = 0, \tag{1}$$

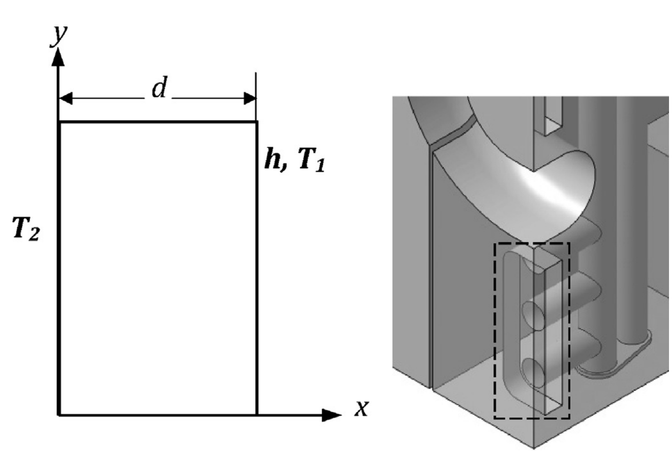

Fig. 2. Convective heat transfer of flat plate with volumetric heat rate (left); example of cover for frontal header of SB, where Model 1 can be applied (right).

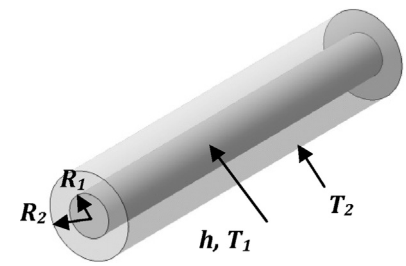

Fig. 3. Convective heat transfer of hollow cylinder with volumetric heat rate.

where Q is volumetric heat rate, t is the temperature along the plate thickness, and λ is thermal conductivity of steel and considered as a constant. This is a steady-state and one dimensional problem.

The boundary conditions are expressed as Eqs. (2) and (3):

$$x = 0, \quad \frac{dt}{dx} = 0 \tag{2}$$

$$x = d, \quad -\lambda\frac{dt}{dx} = h(T_2 - T_1) \tag{3}$$

where h is heat transfer coefficient at the boundary of $x = d$.

Integrating Eq. (1), and then substituting Eqs. (2) and (3) into it, we have

$$\Delta T = T_2 - T_1 = \frac{Qd^2}{2\lambda} + \frac{Qd}{h} \tag{4}$$

It is noticed that the first item in the right-hand of Eq. (4) represents the effect of heat conduction along the plate, and the second item represents the effect of heat convection at the cooling wall.

This model can be applied to the cover for the frontal header of SB, for example, the region marked in rectangle in Fig. 2, where the cross-section of cover can be considered as flat plate.

2.2. Model 2: convective heat transfer in the hollow cylinder

Heat conduction in the hollow cylinder with volumetric heat rate is illustrated as Fig. 3. The heat is removed by the coolant flowing through internal surface of the cylinder, and external surface is assumed as isothermal. The internal and external radius is considered as R_1 and R_2.

The steady-state heat transfer can be described as following:

$$\frac{1}{r}\frac{d}{dr}\left(r\frac{dt}{dr}\right) + \frac{Q}{\lambda} = 0 \tag{5}$$

$$r = R_1, \quad -\left(-\lambda\frac{dt}{dr}\right) = h(t_2 - t_1) \tag{6}$$

$$r = R_2, \quad \frac{dt}{dr} = 0 \tag{7}$$

Integrating Eq. (5) twice, and then substituting Eqs. (6) and (7) into it, the temperature at the external surface can be given by

$$T_2 = C_1 \ln\ r - \frac{1}{4}\frac{Q}{\lambda}r^2 + C_2 \tag{8}$$

$$C_1 = \frac{1}{2}\frac{Q}{\lambda}R_2^2 \tag{9}$$

$$C_2 = \frac{QR_1^2}{4\lambda} - C_1 \ln\ R_1 + T_1 - \frac{QR_1}{2h} + \frac{C_1\lambda}{R_1 h} \tag{10}$$

When h is infinite large, Eq. (10) become:

$$C_2 = \frac{QR_1^2}{4\lambda} - C_1 \ln\ R_1 + T_1 \tag{11}$$