Fusion Engineering and Design xxx (2017) xxx–xxx

Contents lists available at ScienceDirect

Fusion Engineering and Design

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

Thermal-hydraulic design of water cooled first wall of the fusion reactor under DEMO conditions

Phani Domalapally*, Mattia Dellabiancia

Research Centre Řež, Husinec, Řež, Czech Republic

HIGHLIGHTS

- CFD analysis is performed to optimize the geometries of FW cooling channels to handle two different types of heat fluxes.
- For handling normal heat fluxes round tubes with counter current flow are chosen.
- For handling enhanced heat fluxes channel with the Hypervapotron configuration is chosen.
- The final optimized geometry for normal heat fluxes was able to withstand about 1.7 MW/m^2.
- The final optimized geometry for enhanced heat fluxes was able to withstand about 3.2 MW/m^2 heat flux.

ARTICLE INFO

Article history:
Received 3 October 2016
Received in revised form 27 March 2017
Accepted 28 March 2017
Available online xxx

Keywords:
First wall
Heat transfer
Heat flux
Demo

ABSTRACT

The heat loads on the First Wall (FW) of the European DEMO are not yet defined, but when extrapolated from ITER, the loads are expected to exceed the capabilities of current designs. Since DEMO will use Eurofer 97 as the structural material and Pressurized Water Reactor (PWR) conditions at the coolant inlet, the design of the heat sink will be challenging. Indeed, the thermal conductivity of the heat sink material is quite low and the temperature limits are also quite restrictive (between 285 °C and 550 °C). As in ITER, there are two different kinds of heat sinks that were designed for the FW: the first is the normal heat flux channel and the second is the enhanced heat flux channel. For handling normal heat fluxes (NHF), round tubes with counter-current flow are chosen while for handling enhanced heat fluxes (EHF), the Hypervapotron (HV) channel is chosen. Simulations were carried out to find the limits from the thermal hydraulic point of view using commercial Computational fluid dynamics code STAR-CCM+. For NHF channels after optimization the limit from the thermal hydraulics point of view, was found to be ~1.7 MW/m^2, for EHF channels it is ~3 MW/m^2.

1. Introduction

The heat flux load distribution on the FW of the European DEMO reactor is not yet precisely defined. But if the heat loads on the FW are extrapolated from ITER conditions, the values are quite challenging, yet still need to be evacuated. Taking example from ITER, there could be two different kinds of heat sinks for the FW: the first one is Normal Heat Flux (NHF) channel and the second one is Enhanced Heat Flux channel (EHF). The design of the FW itself is challenging as the thermal conductivity ratio of heat sink materials in ITER (CuCrZr) and DEMO (Eurofer 97) is ~10–12. In DEMO the choice of material retained for the FW is governed by:

- Degradation of CuCrZr material properties at irradiation doses >5 dpa.
- Upper temperature limit of CuCrZr does not allow coolant temperatures >350 °C.
- CuCrZr is not a low activation material.

On the other hand the operating conditions of the coolant are PWR conditions ~15.5 MPa, 285–325 °C to allow a contribution of the absorbed FW heat to the DEMO power cycle. However, a lower temperature coolant may be considered if sufficient power absorption is not achievable using PWR water. The DEMO conditions contrast to the 100 °C and 3 MPa in ITER.

For handling normal heat fluxes, round tubes with counter current flow are chosen while for handling enhanced heat fluxes, a channel with the HV configuration is chosen. In the present work simulations are carried out to first, perform optimization of the geometrical configuration of the NHF channels of the FW, in order to

* Corresponding author.
E-mail address: p_kumar.domalapally@cvrez.cz (P. Domalapally).

http://dx.doi.org/10.1016/j.fusengdes.2017.03.164

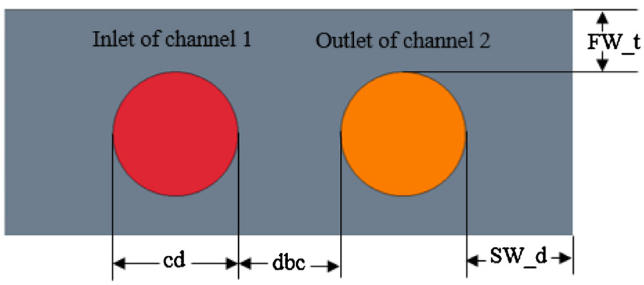

Fig. 1. Initial cross-section of the FW heat sink geometry.

Table 1
Test matrix for NHF channels.

Parameters	Range
Velocity (m/s)	1–8
Front wall thickness (FW_t) (mm)	1–5
Sidewall distance (SW_d) (mm)	1–6
Channel diameter (cd) (mm)	5–13
Distance between the channels (dbc) (mm)	1–8

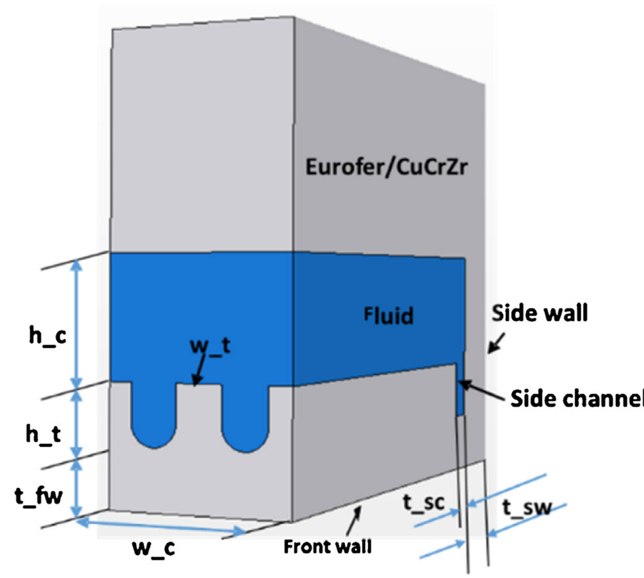

Fig. 2. HV geometry with dimensions.

Table 2
Test matrix for EHF channels.

Parameters	Range(mm)
Thickness of front/side wall (t_fw/t_sw)	1–4
Height of channel (h_c)	2–10
Width of channel (w_c)	12–36
Height of teeth (h_t)	2–6
Width of teeth (w_t)	1.5–4.5
Thickness of side channel (t_sc)	0.5–3

allow the structure to withstand a uniform constant heat flux of at least 0.5 MW/m^2. After finishing the optimization, the maximum heat flux that can be absorbed is estimated. The computational analyses were carried out to keep the maximum temperature limits for the heat sink material below the given limit. In a second step, the performance of the HV heat sink (EHF channel) for potential FW limiter application is investigated under DEMO conditions. Where different, heat fluxes, and geometrical parameters are considered to predict the performance of the HV, the goal of this study is to establish its limits in handling the heat loads before reaching the upper limits from temperature point of view. This work on HV is continuation of the previous work carried out [1] where the geometry was optimized in terms of hydraulic conditions. In order to assess the performance, numerical simulations are performed using CFD code, which was previously validated in predicting the thermal hydraulic performance of HV geometry [1,2].

2. Description

The cross-section of the NHF channel geometry for the analysis is considered from the ITER design [3]. Counter-current flow is used for the analysis based on WCLL Blanket 2014 Design Description Document [4]. The optimization is done by changing the geometrical and hydraulic parameters since the operating conditions are fixed. Fig. 1 shows the initial geometry and Table 1 presents the test matrix used in the analysis.

The dimensions of the channel are 1460 mm in length, 50 mm in width and 20 mm in thickness, with two circular cooling channels. The heat flux of 0.5 MW/m^2 is applied to the top of the heat sink. The channel is designed mainly for the fully decoupled first wall concept, so no heat load is considered from the back side of the channel. Uniform velocity and fully developed flow are set as initial conditions at the inlet. The analysis has been carried out keeping in mind the following restrictions from the mechanical design point of view as per the previous work done at CEA, i.e. the velocity cannot exceed 8 m/s, cd > 5 mm and maximum temperature inside Eurofer <550 °C [5].

The geometry of the HV used for testing its capability is the so-called “Box scraper” geometry [2], which was the most loaded component among all beam line elements used in the JET Neutral Injector Boxes [6]. Apart from this geometry in the previous work several other HV geometries with different cavity shapes and sizes were validated [7], using the CFD where several boiling models were tested. Fig. 2 shows the cross-section of the HV test section used in the current work along with its dimensions. The optimization analysis related to the thermal hydraulic parameters has been carried out and the results are published in [1]. It was mentioned in [1] that in the near future, different geometrical features of the HV are expected to be optimized for performance, and hence the current work is seen in the context of that task. Table 2 show the test matrix for the current optimization.

3. Modeling strategy

In order to perform the simulations for all the geometries, commercial CFD software STAR-CCM+ is used. For NHF channels, only single phase analysis is carried out and the temperature in the fluid was always monitored to ensure it does not exceed the saturation temperature at the given pressure. The numerical scheme used in this paper is based on Navier-Stokes equations. Turbulent flow was accounted for by the Realizable k-ε model. Segregated energy and flow solvers have been applied in the studied models. Periodic boundary conditions are considered on the side wall, as the panel width is not yet determined. For EHF channels the numerical scheme used is also based on the Navier–Stokes equations. Turbulent flow was accounted for by the realizable k–ε model, with wall-functions to handle y+ ~30 near the wall. A pressure-based segregated solver is used for performing the simulations, gravity and surface tension effects are included. Transition boiling model is used to account for boiling in the channels, as was used in the previous work [1,2,7]. The simulations are run until the steady state solution is achieved, where the chosen criterion was the maximum surface temperature in the solid. Before performing the simulations, the following modelling strategy is followed. First of all, the