Fusion Engineering and Design xxx (2015) xxx–xxx

Contents lists available at ScienceDirect

Fusion Engineering and Design

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

Numerical investigation of collector cooling for a 1 MW ITER gyrotron operated with vertical sweeping

Laura Savoldi [a], Cristina Bertani [a], Francesca Cau [b], Fabio Cismondi [b], Gerd Gantenbein [c], Stefan Illy [c], Grazia Monni [a], Roberto Zanino [a,*]

[a] Dipartimento Energia, Politecnico di Torino, Italy
[b] Fusion for Energy, Barcelona, Spain
[c] KIT, Karlsruhe, Germany

ARTICLE INFO

Article history:
Received 12 October 2014
Received in revised form 18 April 2015
Accepted 29 April 2015
Available online xxx

Keywords:
Nuclear fusion
ITER
Gyrotron
Collector
Vertical sweeping
CFD

ABSTRACT

The present gyrotron designs for EC plasma heating in nuclear fusion reactors require the safe exhaust of a power comparable to that injected into the plasma, in order to keep the maximum temperature below the acceptable value of 300 °C. In this paper, the commercial computational fluid dynamics (CFD) software STAR-CCM+® is used to analyze the thermal performance of the annular copper collector of a 1 MW ITER gyrotron, equipped with a hypervapotron structure made of annular fins with rectangular cavities of aspect ratio (depth/width) = 3, cooled by highly subcooled (90–100 °C) pressurized water flowing at ~4 m/s. It is assumed that the simple vertical sweeping strategy is used to reduce the very high peak heat flux on the collector (up to 30 MW/m^2 transient, 5 MW/m^2 time average), due to the spent electron beam. The 2D steady-state conjugate heat transfer problem is solved assuming azimuthal symmetry and accounting for 2-phase flow. The single-cavity flow and heat transfer problem is considered first, to optimize the mesh and the selection of the turbulence model. For the operating conditions considered in this paper, the full collector (100+ cavities) solution shows that boiling occurs only in a limited number of cavities close to the peaks of the heat flux, with the vapor remaining trapped in the bottom of the cavities, i.e. no full hypervapotron regime should be achieved in these operating conditions. The steady-state analysis allows the numerical evaluation of the heat transfer coefficients between Cu and water; these are then used as input for the simplified, purely thermal (solid only) analysis of the actual transient problem for the full collector. The results of the simplified model, which allows a huge reduction of the computational effort, are successfully benchmarked against those of a comprehensive thermal–hydraulic simulation. The computed peak Cu temperature is below the acceptable limit under the steady-state (time averaged) heat load, but becomes unacceptably high if the actual transient heat load is considered, confirming the need for more effective sweeping strategies and/or optimized collector geometries with improved heat transfer capabilities.

1. Introduction

Gyrotrons [1] with 1 MW output power are the key components of the ITER electron cyclotron heating and current drive system [2]. Since the gyrotron efficiency is ~50%, ~1 MW of spent electron beam is to be exhausted by the collector, which is a hollow copper cylinder, cooled by highly sub-cooled water in forced circulation, see Fig. 1a. The water flow has to keep the collector below the maximum acceptable value of 300 °C, notwithstanding the very high heat flux to which it is subject, in order to avoid excessive stresses and the possible damage of the component.

For long time, see [3], hypervapotrons (HVs) have been adopted as a technical solution to the problem of high heat flux exhaust, which besides gyrotrons and their predecessors affect also several other components of fusion reactors, like the divertor [4], the first wall, the ion beam dumps in neutral beam injection systems [5]. Different geometries have been studied including both the flat geometry with a rectangular channel, with or without side slots [6], and the annular geometry [7].

The issue of the high collector heat load can be addressed adopting two combined strategies: (a) equipping the outer collector surface with an annular HV structure, see Fig. 1b, characterized by

* Corresponding author. Tel.: +39 0110904490.
E-mail address: roberto.zanino@polito.it (R. Zanino).

http://dx.doi.org/10.1016/j.fusengdes.2015.04.063
0920-3796/© 2015 Elsevier B.V. All rights reserved.

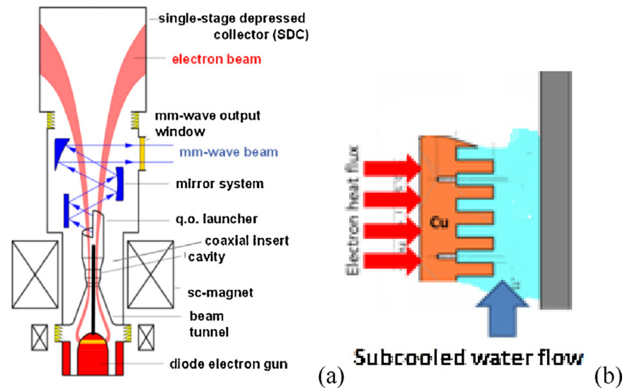

Fig. 1. (a) Main gyrotron sub-assemblies. (b) Collector detail with hypervapotron structure.

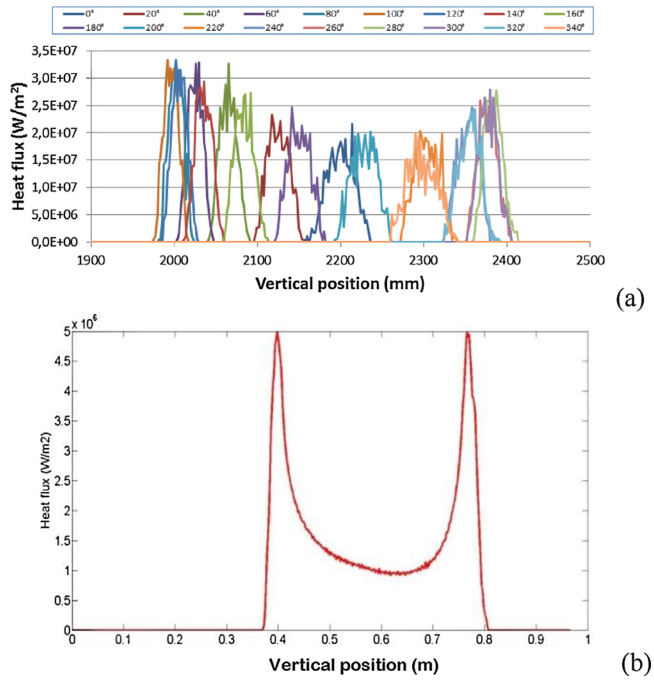

Fig. 2. Spatial distribution of the electron heat flux on the collector surface as computed by the Esray code [14]. (a) Transient distributions at different phases during one cycle. (b) Averaged distribution over one cycle. (The region covered by the teeth shown in Fig. 1b extends from ~0.2 m to ~0.9 m.)

deep rectangular cavities (aspect ratio AR ≡ depth/width = 3); (b) sweeping the spent electron beam onto the collector surface, to more uniformly distribute on it the associated thermal load.

It is well known that the HV thermal–hydraulic performance strongly depends on its geometry and operating conditions (sub-cooling, water flow speed, pressure, . . .). However, as the experimental database available in literature on the HV thermal–hydraulic performance is necessarily limited, computational fluid dynamics (CFD) can be a valuable support to that.

Indeed, in the last years various CFD tools have been used to assess theoretically the HV fluid-dynamics and heat transfer mechanisms [8–12]. These works referred to a flat HV geometry with rectangular flow channel, operated under steady-state heat load conditions, which is of particular interest for plasma-facing components and/or ion beam dumps for neutral beam injectors, whereas here we investigate for the first time with CFD the gyrotron collector problem, characterized by an annular geometry (which justifies the use of 2D models, instead of 3D needed in the case of the flat geometry), subject to transient heat load due to the sweeping of the spent electron beam on the collector surface. The other major new item in the paper is that the comprehensive thermal–hydraulic analysis of the full collector is also used to feed (and benchmark) a simplified, purely thermal model of the solid part of the collector, which allows a significant reduction of the computational cost while still retaining reasonable accuracy in the prediction of the maximum Cu temperature.

The paper is organized as follows: the heat load distribution on the gyrotron collector is described in Section 2; in Section 3 the computational model is presented, which is applied in Section 4 to study at the single-cavity level the steady-state coolant flow and vapor production by boiling; in Section 5 the steady-state thermal–hydraulic analysis of the full collector is presented and heat transfer coefficients deduced from that are input into the transient thermal model of the full collector presented in Section 6.

2. Gyrotron collector geometry and heat load

The wetted (outer) collector surface assumed in the present paper is equipped with a HV structure made of annular fins, with deep rectangular cavities (AR = 3), see Fig. 1b. As very high heat fluxes are expected on the inner collector surface, different strategies for sweeping the spent electron beam are being considered [13]. Here we consider the simplest one, i.e., the so-called vertical sweeping. The transient and time averaged heat flux distributions computed for this case by the Esray code [14], and used as input/driver in the present paper, are shown in Fig. 2a and b, respectively. The sweeping frequency in this case is 7 Hz.

It is seen that the sweeping allows a significant reduction of the transient peak heat flux (up to ~34 MW/m^2), down to ~5 MW/m^2 averaged over one cycle, which is still however not so trivial to handle, since the Cu temperature must be maintained below a maximum acceptable level (300 °C) during operation.

3. Computational model

3.1. Tool

The commercial CFD code Star-CCM+ [15] was chosen, based on previous experience within our group [16] and elsewhere [8–12] on closely related problems, and on difficulties met by different researchers with other commercial codes: Milnes et al. [8] reported convergence problems of ANSYS CFX applied to cavities with AR > 2, while Youchison et al. [10] reported numerical instabilities of Fluent.

3.2. Physics

The conjugate heat transfer problem is solved, with the coolant coupled to the Cu, where a 2D heat conduction problem is solved.

The coolant is made of two phases, liquid and vapor. The volume of fluid (VOF) Eulerian multiphase model [15] is used, assuming the liquid and vapor phases are immiscible and have the same velocity, pressure and temperature. The VOF model includes therefore the conservation laws for a single fluid (mixture of two phases), coupled with an equation describing the transport of the volume fraction of each phase. The latter includes both source terms (vapor formation rate) and diffusive terms, where the turbulent Schmidt number S_{ct}, giving the level of turbulent diffusivity of the bubbles in the liquid core, is required in input.

Under the assumption that the energy transfer between the two phases contributes only to vapor formation (or condensation), the