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 behaviour of the DEMO divertor plasma facing components cooling circuit

P.A. Di Maio [a], S. Garitta [a,*], J.H. You [b], G. Mazzone [c], E. Vallone [a]

[a] University of Palermo, Viale delle Scienze, Edificio 6, 90128 Palermo, Italy
[b] Max Planck Institute of Plasma Physics (E2 M), Boltzmann Str. 2, 85748 Garching, Germany
[c] Department of Fusion and Technology for Nuclear Safety and Security, ENEA C.R. Frascati, via E. Fermi 45, 00044 Frascati, Roma, Italy

HIGHLIGHTS

- Investigation of thermal-hydraulic performances of DEMO divertor cooling system.
- Adoption of a computational fluid-dynamic approach based on finite volume method.
- Comparative study on divertor Plasma Facing Components cooling circuits.
- Assessment of spatial distributions of pressure drop, flow velocity and CHF margin.
- Layout improvements allowing to significantly decrease the total pressure drop.

ARTICLE INFO

Article history:
Received 3 October 2016
Received in revised form 24 January 2017
Accepted 5 February 2017
Available online xxx

Keywords:
DEMO
Divertor
Plasma Facing Components
CFD analysis
Hydraulics

ABSTRACT

Within the framework of the Work Package DIV 1 – “Divertor Cassette Design and Integration” of the EUROfusion action, a research campaign has been jointly carried out by ENEA and University of Palermo to investigate the thermal-hydraulic performances of the DEMO divertor cassette cooling system.

A comparative evaluation study has been performed considering three different options for the cooling circuit layout of the divertor Plasma Facing Components (PFCs). The potential improvement in the thermal-hydraulic performance of the cooling system, to be achieved by modifying cooling circuit layout, has been also assessed and discussed in terms of optimization strategy. The research activity has been carried out following a theoretical-computational approach based on the finite volume method and adopting a qualified Computational Fluid-Dynamic (CFD) code. Results obtained are reported and critically discussed.

1. Introduction

The recent European Fusion Development Agreement roadmap was drafted to realize commercially viable fusion power generation [1]. Within this framework, the divertor is a key in-vessel component, being responsible for power exhaust and impurity removal via guided plasma exhaust. Due to its position and functions, the divertor has to sustain very high heat and particle fluxes arising from the plasma (up to 20 MW/m^2), while experiencing an intense nuclear deposited heat power, which could jeopardize its structure and limit its lifetime. Therefore, attention has to be paid to the thermal-hydraulic design of its cooling system to ensure a uniform and proper cooling, providing a safe margin against Critical Heat Flux (CHF) without an unduly high pressure drop.

In the framework of the activities foreseen by the WP-DIV 1 “Divertor Cassette Design and Integration” of the EUROfusion action [2], a research campaign has been carried out at the University of Palermo, in cooperation with ENEA, to investigate the steady state thermal-hydraulic performances of the DEMO divertor cassette cooling system, focusing the attention on the three different layout options currently under consideration for its Plasma Facing Components (PFCs) cooling circuit [3].

Three separate and independent analyses have been carried out under nominal conditions to evaluate their thermal-hydraulic performances. Specifically, overall coolant thermal rise, overall coolant pressure drop, flow velocity and CHF margin distributions along Plasma Facing Unit (PFU) channels have been assessed, in order to check whether they comply with the corresponding reference limits, namely the maximum coolant total pressure drop (1.4 MPa),

* Corresponding author.
E-mail address: silvia.garitta@unipa.it (S. Garitta).

http://dx.doi.org/10.1016/j.fusengdes.2017.02.025
0920-3796/

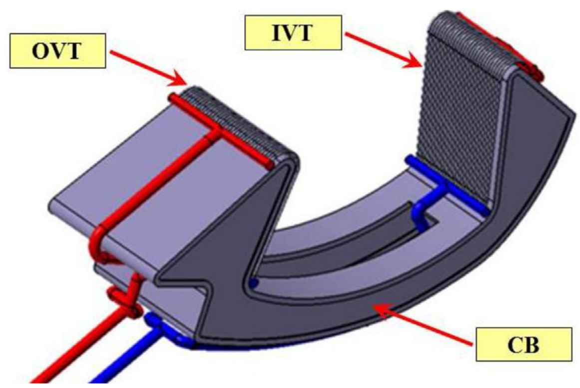

Fig. 1. DEMO divertor cassette 2015 design.

Table 1
Summary of coolant thermal rise calculations.

	Cooling Option 1	Cooling Option 2	Cooling Option 3
Total mass flow rate [kg/s]	60.12	110.22	60.12
ΔT [°C]	9.1	5.0	9.1

the minimum axial flow velocity along PFU channels (16 m/s) and the minimum margin against CHF onset (1.4) at the strike point sections of both Vertical Targets (VTs) PFU channels. Moreover, the assessment of potential layout modifications of the cooling options, allowing the improvement of their thermal-hydraulic performances, has been pursued as a pivotal goal too.

The research campaign has been carried out following a theoretical-computational approach based on the Finite Volume Method and adopting the commercial Computational Fluid-Dynamic (CFD) code ANSYS CFX v.16.2. The analysis models and assumptions are herein reported and critically discussed, together with the main results obtained.

2. Outline of DEMO divertor cassette

According to its 2015 design [2,3], DEMO divertor is articulated in 54 toroidal cassettes, each composed of a Cassette Body (CB) supporting two target plate PFCs, namely an Inner VT (IVT) and an Outer VT (OVT) (Fig. 1), composed of actively cooled PFUs equipped with a Swirl Tape (ST) turbulence promoter.

3. PFCs cooling circuit

Three different layout options (Fig. 2) are currently under investigation for the PFCs cooling circuit [3]. They all rely on the use of subcooled pressurized water at proper inlet pressure and temperature conditions allowing to reach a pressure of 5 MPa and a temperature of 150 °C at VTs strike points [3,4].

In order to assess the thermal-hydraulic performances of each considered cooling option, it has been preliminarily estimated the overall thermal rise experienced by the coolant, under nominal steady state conditions, to remove the PFCs nuclear-deposited heat power reported in [5]. To this purpose, a steady state, isobaric flow has been assumed for the coolant, along with a mass flow rate through each single PFU channel of 1.67 kg/s [2,3]. A follow-up study investigating the potential effects of a reduced mass flow rate combined with a decreased coolant temperature is currently ongoing [6]. Coolant thermal rises have been calculated for the three layout options, hypothesizing water to be at a pressure of 5 MPa and a temperature of 150 °C, and the results obtained are reported in Table 1.

Table 2
Summary of the selected mesh parameters.

	Cooling Option 1	Cooling Option 2	Cooling Option 3
Nodes	$4.97 \cdot 10^{+6}$	$4.78 \cdot 10^{+6}$	$5.33 \cdot 10^{+6}$
Elements	$1.12 \cdot 10^{+7}$	$1.08 \cdot 10^{+7}$	$1.20 \cdot 10^{+7}$
Skewness	0.197	0.202	0.191
Inflation layers number	10	10	10
First layer thickness [μm]	20	20	20
Layers growth rate	1.41	1.41	1.41
Typical element size [m]	$3.08 \cdot 10^{-3}$	$3.48 \cdot 10^{-3}$	$3.60 \cdot 10^{-3}$
Minimum y^+	3.9	4.8	2.8
Average y^+	112.3	141.9	92.9
Maximum y^+	367.9	640.0	3116.6
Model simplification	No ST	No ST	No ST

Table 3
Summary of assumptions, models and BCs.

	Cooling Option 1	Cooling Option 2	Cooling Option 3
Material library	IAPWS IF97	IAPWS IF97	IAPWS IF97
Temperature	150 °C	150 °C	150 °C
Turbulence model	k-ε	k-ε	k-ε
Wall roughness	15 μm	15 μm	15 μm
Inlet BC	p_s = 5 MPa	p_s = 5 MPa	p_s = 5 MPa
Outlet BC	G = 60.12 kg/s	G = 110.22 kg/s	G = 60.12 kg/s

Table 4
PFCs Cooling Option 1 total pressure drops.

	$\Delta p^{(NoST)}$ [MPa]
OVT sub-circuit	1.12
IVT sub-circuit	1.23
TOTAL	1.54

The calculated coolant thermal rises result to be modest, therefore allowing to assume isothermal flow conditions for the PFCs cooling circuit CFD analysis.

4. PFCs cooling circuit CFD analysis

The thermal-hydraulic performances of the three layout options considered for the PFCs cooling circuit have been investigated under nominal conditions by running separate, steady state, isothermal CFD analyses of the flow domains reported in Fig. 2 with the ANSYS CFX v.16.2 code. A summary of the selected mesh parameters and of the main assumptions, models and boundary conditions (BCs) adopted, matured as a further development of similar previous analyses reported [7], is summarized in Tables 2 and 3. A detail of the typical mesh set-up for each CFD analysis is shown in Fig. 3.

4.1. Cooling option 1 CFD analysis results

Total pressure drops numerically assessed across the main sections of PFCs Cooling Option 1 are reported in Table 4.

Since the simplifying hypothesis that no swirl tapes are housed inside the PFU cooling channels has been adopted according to [2], a proper correction of the calculated total pressure drops has to be performed, otherwise they would result heavily underestimated.

To this purpose, a procedure analogous to that used for similar structures in [8,9] has been adopted, conservatively estimating the increase in pressure drop due to STs according to the correlation reported in [10] with reference to the PFU cooling channel where the highest mass flow rate has been numerically predicted. A more detailed description may be found in [2]. As a result, the ST maximum contribution to the total pressure drop amounts to 0.42 MPa,

Please cite this article in press as: P.A. Di Maio, et al., Thermal-hydraulic behaviour of the DEMO divertor plasma facing components cooling circuit, Fusion Eng. Des. (2017), http://dx.doi.org/10.1016/j.fusengdes.2017.02.025