Fusion Engineering and Design 124 (2017) 415–419

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 28 February 2017

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

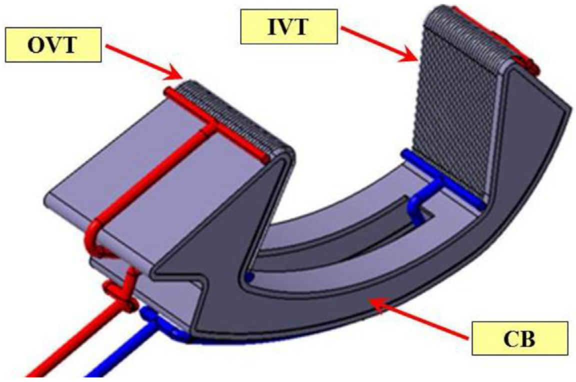

Fig. 1. DEMO divertor cassette 2015 design.

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,

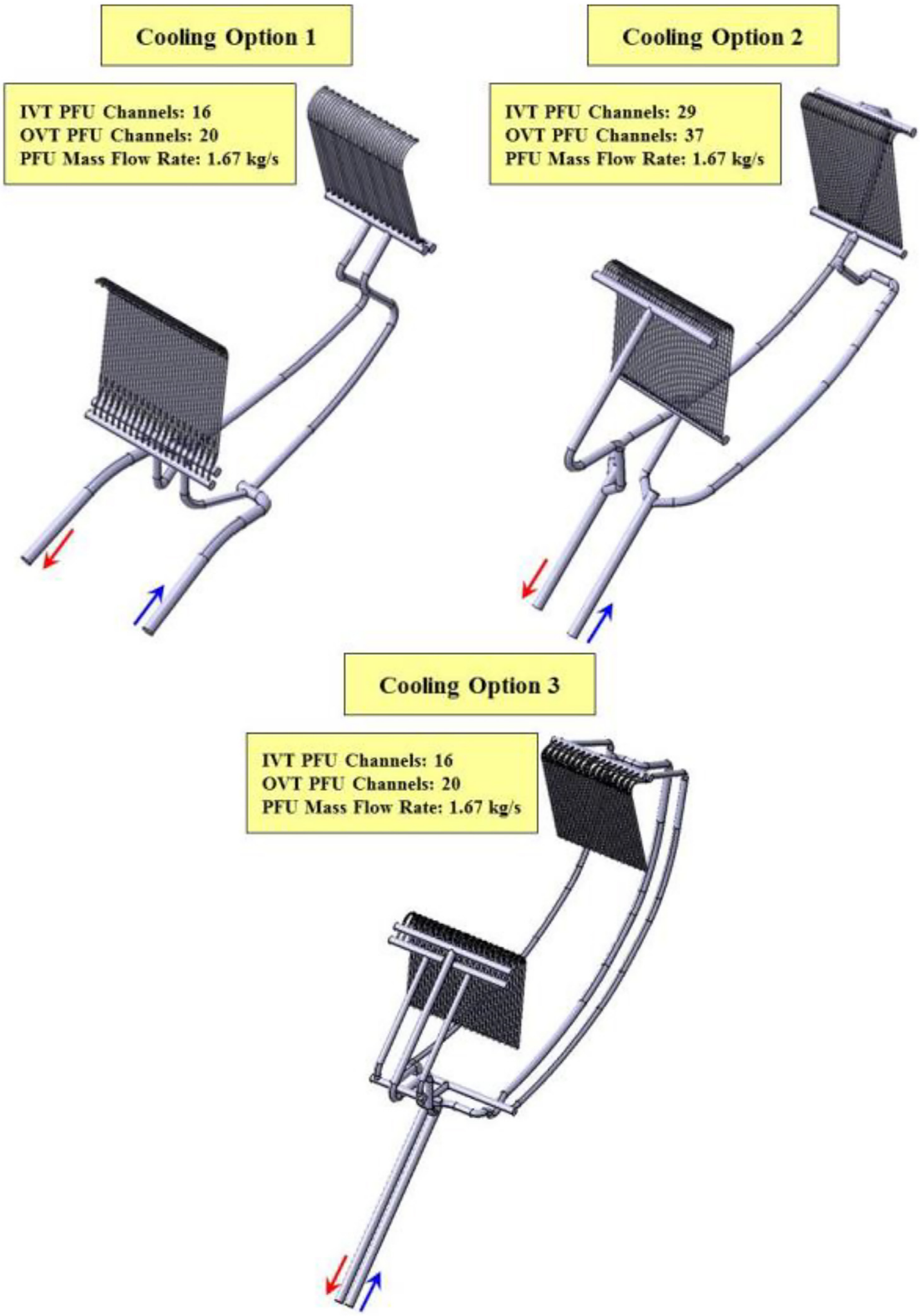

Fig. 2. PFCs cooling circuit options.

Table 5
PFCs Cooling Option 1 VTs mass flow rates.

	Calculated G [kg/s]	Nominal G [kg/s]	Deviation
OVT	32.69	33.40	−2.13%
IVT	27.43	26.72	2.66%

hence the circuit total pressure drop has to be roughly raised to 1.96 MPa, that results to be 0.56 MPa higher than the prescribed limit of 1.4 MPa.

The VTs mass flow rates (Table 5) and the coolant axial flow velocity distribution within PFU channels in presence of STs (Figs. 4 and 5) have been assessed, mainly in order to check whether unbalanced distributions might take place, preventing the uniform and effective cooling of PFUs.

The calculated mass flow rate distribution between VTs is quite acceptable since deviations lower than 3% have been predicted with respect to their nominal values, hence suggesting that VTs hydraulic resistances are properly tuned. Moreover, Figs. 4 and 5 show that, within the VTs PFU channels, the distribution of axial flow velocity is slightly un-uniform, with maximum deviations around 11% and 7% between the maximum and minimum values calculated for OVT and IVT, respectively. However, the predicted minimum velocities for both the OVT (16.8 m/s) and IVT PFU channels (17.7 m/s) result higher than 16 m/s, therefore fulfilling the pertaining requirement.

Finally, the distributions of the margin against CHF onset within the PFU cooling channels of each VT have been assessed, mainly in order to check whether its prescribed minimum value of 1.4 is

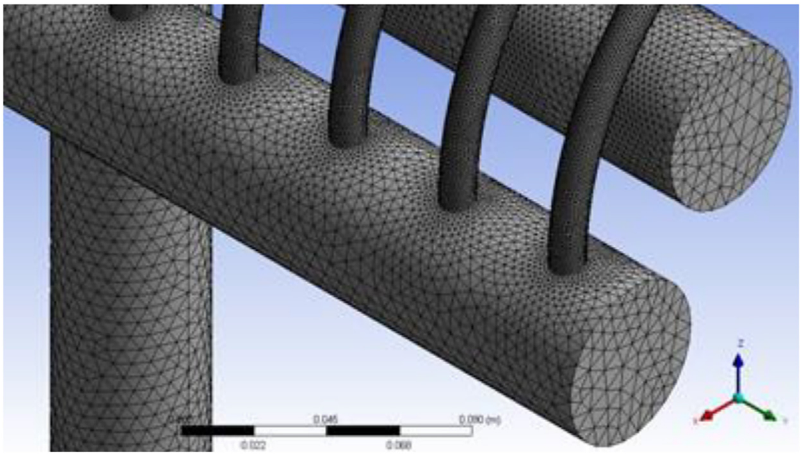

Fig. 3. Detail of a typical mesh set-up.

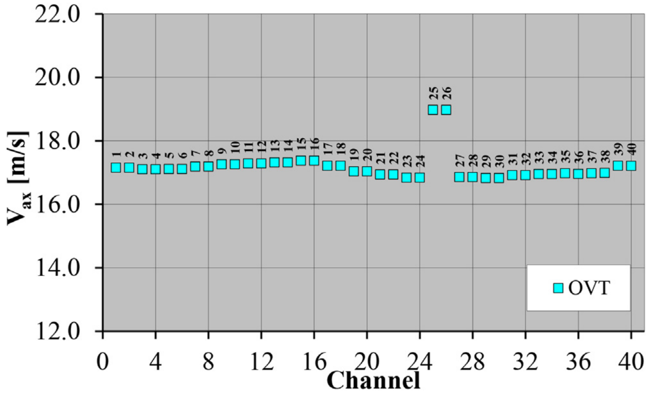

Fig. 4. Axial flow velocities within OVT PFU channels.

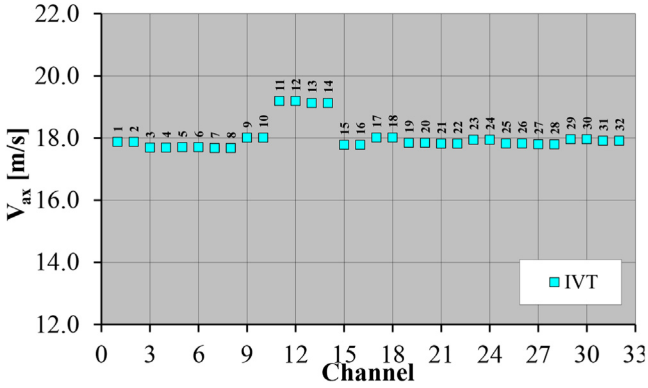

Fig. 5. Axial flow velocities within IVT PFU channels.

guaranteed. To this purpose, attention has been paid to the strike point sections of both OVT and IVT, where it has been supposed to be located the peak value of the incident heat flux arising from plasma, whose intensity has been conservatively assumed equal to 20 MW/m^2. Moreover, the coolant has been supposed to flow, at the pressure of 5 MPa and the temperature of 150 °C, with the calculated axial flow velocity distribution within the PFU channels. In these hypotheses, the CHF at the interface between the coolant and the channel walls has been calculated for each VTs PFU channel by means of the proper correlation given in [10]. Furthermore, assuming a 1.6 peaking factor (f_p), representing the ratio of the maximum heat flux to the coolant to the heat flux incident onto the PFU armour, the corresponding Incident Critical Heat Flux (ICHF) has been assessed as the ratio of the calculated CHF to f_p. Finally, the margin against the CHF onset has been derived within each VTs PFU channel as the ratio of the calculated ICHF to the 20 MW/m^2 nominal heat flux incident onto the PFUs plasma-facing surface. The obtained CHF margin distributions are quite similar to the corresponding axial flow velocity distributions, with minimum and maximum values of 1.55÷1.6 and 1.7, respectively, hence fulfilling the prescribed limit of 1.4. In addition, these results attest an acceptably uniform CHF margin distribution for both the VTs, since maximum deviations between their maximum and minimum values amount to about 8% and 5%, for OVT and IVT respectively.

Table 6
Main results of PFCs cooling circuit CFD analysis.

	Cooling Option 1	Cooling Option 2	Cooling Option 3
G [kg/s]	60.12	110.22	60.12
Δp [MPa]	1.96	2.95	1.84
Min V_{ax} [m/s] OVT	16.83	14.85	14.65
Min V_{ax} [m/s] IVT	17.68	17.38	17.06
Min CHF margin OVT	1.55	1.43	1.41
Min CHF margin IVT	1.61	1.59	1.57

4.2. Summary of CFD analysis results

The results of the CFD analysis carried out for the three layout options under consideration for the PFCs cooling circuit have been summarized in Table 6.

These results suggest that, as far as thermal-hydraulic performances are concerned, the most promising layout seems to be Cooling Option 1. On the other hand, Cooling Option 2 results the most attractive layout, at least from the standpoint of design simplification. Anyway, even if they "substantially" fulfil both axial flow velocity and CHF margin requirements, their total pressure drops have to be further reduced.

4.3. CFD analysis of PFCs revised cooling options

In order to improve the thermal-hydraulic performances of the PFCs cooling circuit, an optimisation study has been carried out, intended to minimize its total pressure drop under nominal steady state conditions. To this purpose, attention has been focussed on both Cooling Options 1 and 2. In particular, considering that a significant portion of the calculated pressure drop is predicted along the PFCs inlet/outlet manifolds, mainly due to distributed hydraulic resistances that strongly depend on hydraulic diameter, it has been investigated the potential effect on pressure drop reduc-

Table 7
Revised Cooling Option 1 total pressure drops.

	$\Delta p^{Reference}$ [MPa]	$\Delta p^{Revised}$ [MPa]	Δ [MPa]
OVT sub-circuit	1.12	0.92	0.20
IVT sub-circuit	1.23	0.87	0.36
TOTAL	1.54	1.02	0.52

Table 8
Revised Cooling Option 2 total pressure drops.

	$\Delta p^{Reference}$ [MPa]	$\Delta p^{Revised}$ [MPa]	Δ [MPa]
OVT sub-circuit	1.63	0.92	0.71
IVT sub-circuit	1.84	0.98	0.86
TOTAL	2.52	1.16	1.36

tion of a manifold diameter increase by a factor 1.3. The pressure drops across the investigated circuits main sections are reported in Tables 7 and 8.

The CFD numerical predictions have been corrected for the STs presence and, hence, the total pressure drops have been roughly raised to $\Delta p = 1.02 + 0.39 = 1.41$ MPa, as to the revised Cooling Option 1, and to $\Delta p = 1.16 + 0.31 = 1.47$ MPa, as to the revised Cooling Option 2, obtaining in both cases values slightly higher than the prescribed limit of 1.4 MPa that clearly encourage to pursue the manifold diameter increase to improve PFCs cooling circuit hydraulic performances.

5. Conclusions

Within the framework of the activities foreseen in the WPDIV of the EUROfusion Consortium, a computational study has been carried out at the University of Palermo, in cooperation with ENEA, to investigate the steady state thermal-hydraulic performance of three different design options of cooling circuit layout for the DEMO divertor target plate PFCs.

Results obtained have indicated only modest temperature rise of water coolant (< 10 °C) for all investigated target plate cooling options and slight margin against the CHF, being slightly above the prescribed criterion of 1.4. However, the estimated total pressure drop turned out to be higher than the specified limit of 1.4 MPa and the limit on the minimum axial flow speed could not always be reached, while the local maximum flow speed seemed too high to be adopted.

On the other hand, the revised pipework configurations led to significant decrease in the predicted total pressure drop, thus encouraging a further layout optimization to enhance thermal-hydraulic performance.

A further CFD simulation campaign is ongoing, where the impact of reduced coolant inlet temperature on the overall thermal-hydraulic behaviour (e.g. coolant speed, CHF margin) is investigated.

Acknowledgments

This work has been carried out within the framework of the EUROfusion Consortium and has received funding from the Euratom research and training programme 2014-2018 under grant agreement No 633053. The views and opinions expressed herein do not necessarily reflect those of the European Commission.

References

[1] F. Romanelli et al., Fusion Electricity – A Roadmap to the Realisation of Fusion Energy, European Fusion Development Agreement (EFDA) (2012), ISBN 978-3-00-040720-8T.
[2] Final Report on Deliverable DEMO Divertor – Thermo-hydraulic assessmentreport 2015, Report IDM reference No EFDA_D_2MY45W, DIV-1-T001-D010.
[3] J.H. You, et al., Conceptual design studies for the European DEMO divertor: rationale and first results, Fusion Eng. Des. 109–111 (2016) 1598–1603.
[4] J.H. You et al., European DEMO divertor target: Operational requirements and material-design interface, Nuclear Materials and Energy, doi:http://dx.doi.org/10.1016/j.nme.2016.02.005 in press.
[5] C. Bachmann et al., Issues and strategies for DEMO in-vessel component integration, Fusion Eng. Des., doi:http://dx.doi.org/10.1016/j.fusengdes.2016.05.040 in press.
[6] EUROfusion, personal communications.
[7] P.A. Di Maio, et al., On the hydraulic behaviour of ITER Shield Blocks #14 and #08. Computational analysis and comparison with experimental tests, Fusion Eng. Des. 109–111 (2016) 30–36.
[8] P.A. Di Maio, et al., Numerical simulation of the transient thermal-hydraulic behaviour of the ITER blanket cooling system under the draining operational procedure, Fusion Eng. Des. 98–99 (2015) 1664–1667.
[9] P.A. Di Maio, et al., Analysis of the steady state hydraulic behaviour of the ITER blanket cooling system, Fusion Eng. Des. 98–99 (2015) 1470–1473.
[10] A.R. Raffray, et al., Critical heat flux analysis and R&D for the design of the ITER divertor, Fusion Eng. Des. 45 (1999) 377–407.