Analysis of the steady state hydraulic behaviour of the ITER blanket cooling system

P.A. Di Maio[a*], G. Dell'Orco[b], A. Furmanek[b], S. Garitta[a], M. Merola[b], R. Mitteau[b], R. Raffray[b], G.A. Spagnuolo[a], E. Vallone[a]

[a] *Dipartimento di Energia, Ingegneria dell'Informazione e Modelli Matematici, Università di Palermo Viale delle Scienze, 90128, Palermo, ITALIA*
[b] *ITER Organization, Route de Vinon-sur-Verdon, CS 90 046, 13067 St Paul Lez Durance Cedex, FRANCE*

The blanket system is the ITER reactor component devoted to providing a physical boundary for plasma transients and contribute to thermal and nuclear shielding of vacuum vessel, magnets and external components. It is expected to be subjected to significant heat loads under nominal conditions and its cooling system has to ensure an adequate cooling, preventing any risk of critical heat flux occurrence while complying with pressure drop limits.

At the University of Palermo a study has been performed, in cooperation with the ITER Organization (IO), to investigate the steady state hydraulic behaviour of the ITER blanket standard sector cooling system. A theoretical-computational approach based on the finite volume method has been followed, adopting the RELAP5 system code. Finite volume models of the most critical blanket cooling circuits have been set-up, realistically simulating the coolant flow domain. The steady state hydraulic behaviour of each cooling circuit has been investigated, determining its hydraulic characteristic function and assessing the spatial distribution of coolant mass flow rates, velocities and pressure drops under reference nominal conditions.

Results obtained have indicated that the investigated cooling circuits are able to provide an effective cooling to blanket modules, generally meeting ITER requirements in term of pressure drop and velocity distribution, except for a couple of circuits that are being revised.

Keywords: ITER; blanket; hydraulics.

1. Introduction

The blanket system represents one of the pivotal components of the ITER reactor, providing a physical boundary for the plasma transients and contributing to the thermal and nuclear shielding of the vacuum vessel, the superconducting magnets and the external ITER components. It is composed of 440 modules, connected to the vacuum vessel through a mechanical attachment system of flexible supports and keys and distributed in 18 toroidal sectors, covering a plasma-facing surface of ~650 m^2 [1]. From the structural standpoint, a typical blanket module is ~1 m high in poloidal direction, ~1.5 m long in toroidal direction and ~0.5 m thick in radial direction. It is composed of a plasma-facing First Wall (FW) panel and a Shield Block (SB), both actively cooled by pressurized water fed by a system of inlet/outlet manifolds connected to the Integrated Blanket, Edge Localized Mode Coil and Divertor (IBED) Primary Heat Transfer System (PHTS) of the ITER TOKAMAK Cooling Water System (TCWS). These manifolds are designed to feed the proper coolant mass flow rate to each blanket module, thus allowing the extraction of the 736 MW of maximum thermal power deposited inside the blanket under reference nominal conditions.

As a consequence of its position and functions, the blanket system will be subjected to significant heat loads under nominal reference conditions, namely a surface heat flux on the FW, due to radiation and particle fluxes from the plasma, and a volumetric heating from the neutron energy deposition in the blanket. Therefore, the design of its cooling system can be particularly demanding since it has to ensure that adequate cooling is provided to each module to prevent any risk of critical heat flux occurrence while complying with ITER pressure drop limits to avoid an unacceptably high pumping power.

An intense analysis campaign has been performed at the Department of Energy, Information Engineering and Mathematical Models (DEIM) of the University of Palermo in close cooperation with the ITER Organization (IO) to investigate the thermal-hydraulic behaviour of the cooling system of a standard 20° ITER blanket sector. The analysis has been performed following a theoretical-computational approach, based on the adoption of the finite volume method, and it has been carried out by means of the RELAP5 Mod3.3 system code [2], widely qualified for the numerical simulation of thermal-hydraulic transients in light water nuclear fission reactors and already successfully adopted at the DEIM for the numerical modelling of the thermal-hydraulic behaviour of the divertor cassette, the Upper Port Plug Trapezoid Section, the Test Blanket Module Port Plug, both under steady state and transient draining conditions [3,4,5].

The present paper summarizes the research activity, describing the modelling strategies adopted for the blanket cooling system investigation and critically discussing the results obtained as to its steady state hydraulic behaviour.

author's email: pietroalessandro.dimaio@unipa.it

2. ITER blanket standard sector cooling system

The generic 20° ITER blanket sector is composed of 24 modules distributed along the poloidal direction in the inboard, upper and outboard segments and along the toroidal direction in the central (C) or side (S) regions.

It is connected to a cooling system composed of 20 independent hydraulic circuits connected in parallel to the Upper Ring Manifold (URM) sub-headers of TCWS IBED PHTS. In particular, 16 cooling circuits individually cool 16 single modules, while the remaining 4 circuits provide cooling to 4 couples of twinned modules and specifically modules 6-7, 10S-11S, 12C-13C and 12S-13S. This cooling system is designed to feed a proper mass flow rate of sub-cooled water at 70°C and 4 MPa to each circuit, allowing heat power deposited inside blanket modules to be extracted with a 71.6°C maximum thermal rise while complying with ITER requirements in term of maximum in-vessel and circuit pressure drops, set to 1.35 and 1.6 MPa, respectively. A cooling circuit of the blanket standard sector cooling system is composed of a couple of inlet/outlet manifolds, a coaxial connector and a blanket module cooling circuit.

2.1. Manifolds

Manifolds are devoted to independently and separately route water coolant from the inlet URM sub-header of TCWS IBED PHTS to either a single blanket module or a couple of twinned modules, routing it back to the corresponding outlet URM sub-header (Fig. 1). They are subdivided, in correspondence to the chimney bulkhead, into an in-vessel and an out-vessel segment.

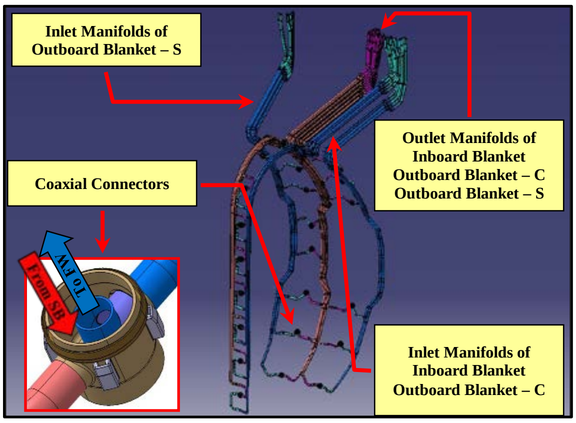

Fig. 1. Blanket standard sector in-vessel manifolds.

2.2. Coaxial connectors

Coaxial Connectors (CC) are intended to allow the hydraulic connection between inlet/outlet manifolds and blanket module cooling circuits, each of them being composed of an internal elbow that receives coolant from the inlet manifold and routes it to the FW cooling circuit, and an external cylindrical jacket housing the elbow, that receives water from the SB cooling circuit and routes it to the outlet manifold (Fig. 1).

2.3. Blanket module cooling circuits

Each blanket module is designed with a circuit ensuring adequate cooling to prevent any risk of critical heat flux occurrence while complying with pressure drop limits. It is composed of FW and SB cooling circuits (Figs. 2, 3), connected in series by means of flexible pipes at the FW-SB interface. Each circuit has a lay-out compatible with the heat load to be extracted under reference nominal conditions. In particular, the FW circuit is based on the adoption of separate Plasma Facing Unit (PFU) cooling circuits (Fig. 4), made up of either hypervapotrons or circular channels, connected in parallel, while the SB circuit consists of a complex network of gun-drilled cooling channels and plates connected in parallel to toroidal headers. Further details may be found in [1,6].

Water coolant coming from the inlet manifold is routed by the coaxial connector to the FW cooling circuit, from which it is passed, through flexible pipes, to the SB cooling circuit, before being routed to the outlet manifold flowing through the coaxial connector.

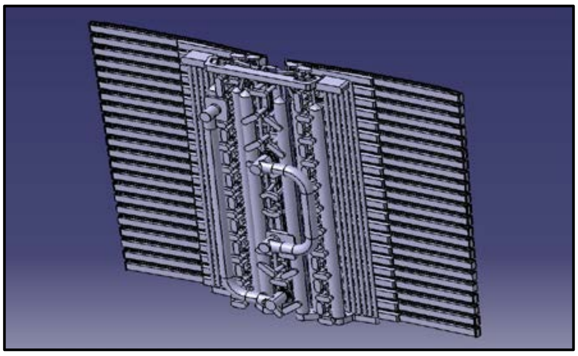

Fig. 2. Typical lay-out of FW cooling circuit.

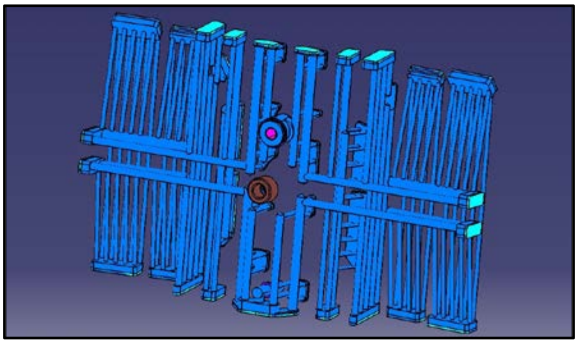

Fig. 3. Typical lay-out of SB cooling circuit.

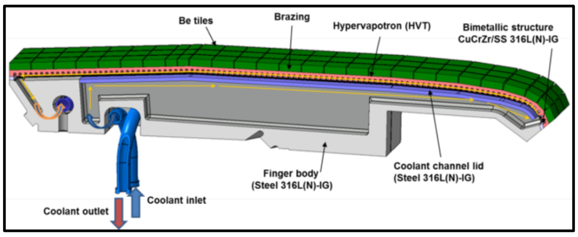

Fig. 4. Typical lay-out of FW PFU cooling circuit.

3. Hydraulic analysis

A complete hydraulic analysis using the RELAP5 system code has been carried out for each one of the most demanding cooling circuits of the generic 20° ITER blanket sector cooling system.

3.1. Model set-up

A realistic finite-volume model of each investigated cooling circuit has been set-up, consisting of a geometrical, a constitutive and a hydraulic model.

The geometrical model simulates in a quasi-2D approach the spatial features of the flow domain of each cooling circuit to be investigated, including its inlet and outlet manifolds delimited by the relevant URM sub-headers (Fig. 5). Upstream and downstream circuits have been modelled by fixed-pressure volumes. The discretization has been performed saving the volume of each component, so to realistically simulate the overall coolant amount. Moreover, sub-volumes have been properly oriented in space in order to reproduce their relative positions and heights.

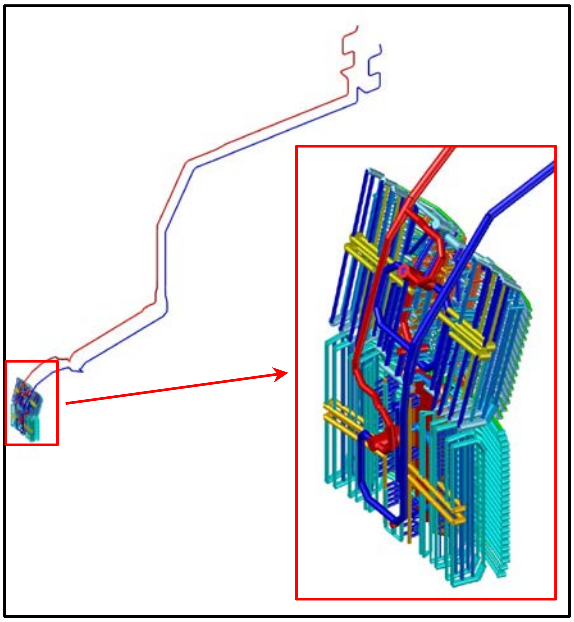

Fig. 5. Geometrical model of modules 6-7 cooling circuit.

The constitutive model provided by the RELAP5 code has been adopted to simulate the thermo-dynamic behaviour of water coolant by proper libraries that take into account the dependence of its thermo-physical properties on pressure, temperature and quality.

The hydraulic model has been set-up to properly simulate water single-phase flow inside the investigated cooling circuits. Concentrated hydraulic resistances have been modelled by the following equation:

$$\Delta p = K\rho\frac{v^2}{2} \quad (1)$$

where p represents the total pressure, v the flow average velocity and K the concentrated loss factor, whose values have been drawn from [7], according to specific geometrical configurations and flow conditions. Further details are reported in [6]. Distributed hydraulic resistances have been modelled by the well-known equation:

$$\Delta p = \lambda\frac{L}{D_H}\rho\frac{v^2}{2} \quad (2)$$

where D_H is the hydraulic diameter and λ the well-known Darcy’s friction factor, calculated by RELAP5 code according to flow regime and absolute roughness (δ) adopting the Zigrang and Sylvester correlation [2]. To this purpose, a proper spatial distribution has been agreed with IO, that foresees δ equal to 50 μm for drilled channel and out-vessel manifolds, 15 μm for in-vessel manifolds and coaxial connectors and 25 μm for FW channels. Further details, particularly those relevant to hypervapotron resistance modelling, can be found in [6].

Since both concentrated and distributed hydraulic resistances depend on the spatial distribution of the flow velocity field, it has been necessary to perform iterative analyses to properly take into account this dependence.

3.2. Steady state analysis

The nominal steady state hydraulic behaviour of each circuit considered has been investigated with a focus on the assessment of both pressure drops and velocity spatial distributions mainly in the FW PFU channels. In any case isothermal flow of water coolant at 371 K has been assumed.

Furthermore, the steady state hydraulic behaviour of each module cooling circuit has been characterized by assessing its characteristic function, $\Delta p = \Delta p(G)$, which gives the functional dependence of the pressure drop across inlet/outlet coaxial connector, Δp, onto the corresponding mass flow rate, G, under steady state conditions.

To this purpose a set of parametric analyses has been launched for each module cooling circuit under the assumption of isothermal water flow at 371 K induced by an imposed pressure drop (Δp_i) ranging typically from 0.1 to 1 MPa to evaluate the relevant mass flow rate (G_i). The hydraulic characteristic functions have been obtained by best fitting the (G_i, Δp_i) pairs calculated with the following power-law analytical form:

$$\Delta p(G) = \alpha G^{\beta} \quad (3)$$

where α and β coefficients depend on water temperature; they have been numerically assessed when the pressure drop is expressed in MPa and the mass flow rate in kg/s.

3.3. Results

As far as nominal steady state hydraulic analyses are concerned, the pressure drop distributions calculated within each circuit investigated, together with the data relevant to the in-vessel segment and the module cooling system, are summarized in Table 1.

Table 1. Pressure drop distribution.

Cooling circuit		G [kg/s]	Δp [MPa]		
			Total	In-vessel	Module
#1		5.400	0.851	0.633	0.464
#2		5.542	0.896	0.652	0.488
#6-7		11.039	0.861	0.645	0.482
	#6	5.299	-	-	0.450
	#7	5.740	-	-	0.435
#10S		6.709	0.554	0.463	0.334
#12C		5.056	0.343	0.287	0.237
#12S		5.056	0.338	0.292	0.237
#18S		6.526	1.025	0.705	0.550

Pressure drops calculated have been compared to the pertaining ITER limits, concluding that an adequate cooling is ensured for the ITER blanket standard sector cooling system without an unduly high pumping power.

Furthermore, velocity distributions calculated within FW PFU channels of the considered cooling circuits are summarized in Table 2, indicating that symmetric and quite acceptable velocity profiles are generally predicted.

Finally, as to the hydraulic characteristic functions of the investigated module cooling circuits, the calculated values of their coefficients are reported in Table 3.

Table 2. FW PFU channel velocity distribution [m/s].

Cooling circuit	G [kg/s]	Left wing	Right wing	$\frac{V_{Left} - V_{Right}}{V_{Left}}$
		V_{Left}	V_{Right}	
#1	5.400	2.257	2.333	-0.034
#2	5.542	2.314	2.393	-0.034
#6	4.684	1.956	2.022	-0.034
#7	6.355	2.014	2.414	-0.198
#10S	6.709	3.009	3.067	-0.019
#12C	5.056	1.987	2.042	-0.028
#12S	5.056	1.987	2.042	-0.028
#18S	6.526	1.259	1.418	-0.126

Table 3. Hydraulic characteristic functions coefficients.

Module cooling circuit	Coefficients	
	α	β
#1	0.0160	1.9957
#2	0.0160	1.9957
#6	0.0160	1.9957
#7	0.0127	1.9995
#10S	0.0076	1.9922
#12C	0.0095	1.9884
#12S	0.0095	1.9884
#18S	0.0130	1.9962

4. Conclusions

A theoretical-computational analysis campaign has been performed by DEIM and IO to investigate the hydraulic behaviour of the ITER blanket standard sector cooling system under nominal steady state conditions.

Results have confirmed the aptitude of this cooling system to remove nuclear deposited power from blanket modules with an acceptable pumping power.

Disclaimer

The views and opinions expressed herein do not necessarily reflect those of the ITER Organization.

References

[1] A. R. Raffray et alii, The ITER Blanket System Design Challenge, Nuclear Fusion 54 (2014) 033004.

[2] RELAP5 Mod3.3 code - Theory Manual, INEL Nuclear Safety Analysis Division.

[3] A. Tincani, P. A. Di Maio, G. Dell'Orco, I. Ricapito, B. Riccardi, G. Vella, Steady state and transient thermal-hydraulic characterization of full-scale ITER divertor plasma facing components, Fusion Engineering and Design 83 (2008) 1034-1037.

[4] P. A. Di Maio, D. Paradiso, G. Dell'Orco, C.S. Pitcher, M. Kalish, On the theoretical–numerical study of the ITER Upper Port Plug structure hydraulic behaviour under steady state and draining and drying transient conditions, Fusion Engineering and Design 86 (2011) 2983-2998.

[5] B. Y. Kim et alii, Status of ITER TBM port plug conceptual design and analyses, Fusion Engineering and Design 89 (2014) 1969-1974.

[6] ITER private communication, ITER_D_NPFHH8 v1.0.

[7] I. E. Idelchik, Handbook of Hydraulic Resistance, 1986.