Numerical simulation of the transient thermal-hydraulic behaviour of the ITER blanket cooling system under the draining operational procedure

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*

Within the framework of the research and development activities supported by the ITER Organization on the blanket system issues, an intense analysis campaign has been performed at the University of Palermo with the aim to investigate the thermal-hydraulic behaviour of the cooling system of a standard 20° sector of ITER blanket during the draining transient operational procedure.

The analysis has been carried out following a theoretical-computational approach based on the finite volume method and adopting the RELAP5 system code. In a first phase, attention has been focused on the development and validation of the finite volume models of the cooling circuits of the most demanding modules belonging to the standard blanket sector. In later phase, attention has been put to the numerical simulation of the thermal-hydraulic transient behaviour of each cooling circuit during the draining operational procedure. The draining procedure efficiency has been assessed in terms of both transient duration and residual amount of coolant inside the circuit, observing that the former ranges typically between 40 and 120 s and the latter reaches at most ~8 kg, in the case of the cooling circuit of twinned modules #6-7. Potential variations to operational parameters and/or to circuit lay-out have been proposed and investigated to optimize the circuit draining performances. In this paper, the set-up of the finite volume models is briefly described and the key results are summarized and critically discussed.

Keywords: thermal-hydraulic, RELAP5, draining, blanket.

1. Introduction

An intense analysis campaign has been launched by 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 performances of the cooling system of a standard 20° ITER blanket sector under nominal steady state conditions as well as during the Draining and Drying (D&D) transient operational procedure.

In particular, the activity has been mainly focussed on the investigation of the cooling system aptitude to safely and timely undergo a D&D transient operational procedure, carried out according to the technical specifications of the ITER Tokamak Cooling Water System (TCWS) D&D system [1].

The activity has been performed following a theoretical-computational approach, based on the adoption of the finite volume method, and carried out by means of the RELAP5 Mod3.3 system code [2]. This code is widely qualified for the numerical simulation of thermal-hydraulic transients in light water nuclear fission reactors and it has already been 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 draining transient conditions [3,4,5].

2. Blanket standard sector cooling system

The standard 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 separate and independent hydraulic circuits connected in parallel to the Upper Ring Manifold (URM) sub-headers of the ITER TCWS Integrated Blanket, Edge Localized Mode Coils and Divertor (IBED) Primary Heat Transfer System (PHTS). In particular, 16 cooling circuits individually cool 16 single modules, while the remaining 4 circuits provide cooling to 4 couples of twinned modules, 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 temperature rise while complying with ITER requirements in terms 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.

A more detailed description of the blanket standard sector cooling system, of its lay-out and operating conditions may be found in [1,6].

author's email: eug.vallone@gmail.com

3. Research campaign

The analysis activity has been focused on investigating the thermal-hydraulic behaviour of the blanket standard sector cooling system during the D&D transient operational procedure, to be carried out prior to each maintenance and or inspection operation for each blanket module by the injection of high pressure nitrogen.

Attention has been focussed on the cooling circuits of the most critical and demanding modules of the standard blanket sector to assess for each one of them its aptitude to be completely and quickly drained according to the envisaged operational procedure, determining its draining performances mainly in terms of residual coolant amount at the operational transient end. Furthermore, potential variations to the procedure operational parameters (nitrogen pressure) and/or to the circuits lay-outs (valves inertia) have been proposed and investigated in view of the improvement and optimization of the cooling system draining performances.

To this purpose, transient thermal-hydraulic analyses have been carried out for each one of the considered cooling circuits, using the RELAP5 system code.

3.1. Model set-up

According to the requirements of RELAP5 code family, a realistic finite-volume model of each of the considered cooling circuits of the blanket standard sector cooling system has been set-up, consisting of a geometrical, a constitutive and a hydraulic model.

The geometrical model realistically simulates in a quasi-2D approach the flow domain of each cooling circuit to be investigated, including its inlet and outlet manifolds delimited by the relevant URM sub-headers. Upstream and downstream circuits have been modelled by fixed-pressure volumes. The discretization has been performed saving both the volume of each component, to realistically simulate the overall coolant amount, and its orientation in space to reproduce relative positions and heights. The constitutive models provided by the RELAP5 code have been adopted to describe the thermo-dynamic behaviour of fluids (water and/or nitrogen) circulating inside the cooling circuits. As to water coolant, the model is based on proper libraries that describe the dependence of its thermo-physical properties on pressure, temperature and, possibly, quality. Concerning nitrogen, the ideal gas model has been adopted. The hydraulic model simulate single-phase water flow and/or two-phase water-nitrogen flow inside the cooling circuits being investigated. The concentrated and distributed hydraulic resistances occurring within the flow domain have been modelled considering their possible functional dependence on flow velocity field.

The models have been assessed investigating the steady state thermal-hydraulic behaviour of the considered cooling circuits. Their description together with a critical discussion of the steady state results obtained may be found in [6,7].

3.2. Draining transient procedure

The draining procedure is carried out in two subsequent phases, namely the gravity draining and the forced draining. During the gravity draining, water inside FW/BLK PHTS components and pipelines is drained by gravity, partially evacuating the components. At the same time initial gas charging of the TCWS Draining System takes place. During the forced draining phase, the Draining System compressor starts to inject nitrogen gas inside the circuit to blow-out the remaining water within the component. In particular, nitrogen is first pressurized up to 4 MPa (reduction to 2 MPa is under consideration), then cooled by a condenser to remove its potential residual humidity, subsequently heated by an electric heater up to 50 °C and it is finally distributed to the proper PHTS loop, which routes it to the component to be drained [7].

3.3. Draining transient analyses

During the draining procedure, complex thermal-hydraulic phenomena arise since a two-component (water-nitrogen), two-phase (liquid-gas) flow occurs inside a significant portion of the circuit, that progressively changes in time. Transient thermal-hydraulic analyses have been performed to investigate these phenomena also in view of the D&D procedure optimization, which aims to minimize the amount of water remaining inside the circuit, maximizing the coolant mass discharged and minimizing the transient duration. The analyses have been carried out separately for each cooling circuit of the blanket standard sector cooling system considered in the study, in order to compare their performances manly in terms of residual water and transient duration.

According to the draining procedure, a realistic scenario for the draining operational transient has been considered, assuming nitrogen pressurized at 4.1 MPa injected at 50 °C inside each cooling circuit, full of water at 50 °C and 0.1 MPa. The potential effects of the reduction of nitrogen injection pressure down to 2.1 MPa or even to 1.1 MPa have been investigated for critical cooling circuits, whose draining potential has been considered insufficient due to their position and/or lay-out. The gravity draining phase has been neglected, since previous analyses have shown that it has no pivotal role in the draining procedure, and attention has been focussed on the forced draining phase. The Henry-Fauske model, already implemented and validated in the RELAP5 code, has been adopted to realistically simulate the potential onset of choked flow conditions inside the cooling circuit during the transient [8].

The time evolutions of both nitrogen and water mass flow rates have been numerically evaluated at the inlet and outlet of each cooling circuit, allowing the total nitrogen mass injected into the circuit and, especially, the total amount of water discharged to be calculated. In particular, since calculated mass flow rates may be affected by minor numerical errors whose effects may become not negligible when they are integrated over time and in order to have a more accurate estimate of the residual water amount at the end of the transient, the evaluation has been made on the basis of the final distribution of the averaged liquid density and void ratio in over all model volumes. Further details may be found in [7].

The transient has been considered to end as soon as the total inlet mass flow rate equals the outlet one. Finally, in order to assess the draining performances of each cooling circuit investigated, the following draining efficiency parameter has been introduced:

$$\varepsilon = \frac{M_{l0} - M_{l}^{residual}}{M_{l0}} \tag{1}$$

where M_{l0} is the cooling circuit initial water amount.

3.4. Results

The transient thermal-hydraulic behaviour of the considered cooling circuits under the forced draining operational procedure has been numerically investigated, taking into account two different operational scenarios.

On a first phase, the so-called reference draining (RD) scenario, lasting for 60 s, has been simulated according to the previously described assumptions. In a second phase, a complete draining (CD) scenario has been simulated taking into account the presence of an inlet globe valve with an opening time of 20 s and of a similar outlet valve, stopping nitrogen flow, in order to study their potential effects on the cooling circuits draining performances mainly in terms of transient duration and residual water distribution.

Main results obtained are summarized in Tables 1-4.

Table 1. RD scenario: residual water distribution.

Cooling circuit	p_{in} [MPa]	Residual water [kg]				
		Total	Manifolds	Module		
				Total	FW	SB
#1	4.1	2.599	0.003	2.596	2.327	0.269
#2	4.1	2.589	0.004	2.586	2.320	0.266
#6-7	4.1	7.810	0.861	-	-	-
#6	-	-	-	5.734	4.790	0.944
#7	-	-	-	1.215	0.351	0.864
#6-7	2.1	8.108	0.592	-	-	-
#6	-	-	-	6.524	5.209	1.314
#7	-	-	-	0.992	0.260	0.732
#10S	4.1	3.389	2.666	0.723	0.228	0.496
#12C	2.1	5.253	4.568	0.685	0.252	0.433
#12S	2.1	5.432	4.749	0.683	0.250	0.433
#15S	4.1	1.946	0.756	1.190	0.981	0.209
#16C	2.1	3.927	0.191	3.735	0.243	3.492
#17C	4.1	0.019	0.001	0.018	0.001	0.017
#18S	4.1	0.161	0.001	0.160	0.010	0.150

Table 2. CD scenario: residual water distribution.

Cooling circuit	p_{in} [MPa]	Residual water [kg]				
		Total	Manifolds	Module		
				Total	FW	SB
#1	4.1	2.569	0.006	2.563	2.305	0.258
#2	4.1	2.560	0.007	2.553	2.297	0.256
#10S	4.1	3.385	2.665	0.720	0.234	0.485
#12C	2.1	5.253	4.576	0.678	0.258	0.420
#12S	2.1	5.424	4.745	0.679	0.259	0.420
#15S	4.1	1.262	0.702	0.560	0.431	0.129
#16C	2.1	3.765	0.195	3.571	0.405	3.166
#17C	4.1	0.015	0.005	0.010	0.001	0.009
#18S	4.1	0.157	0.002	0.155	0.010	0.145

Table 3. RD scenario: draining efficiency.

Cooling circuit	p_{in} [MPa]	Draining efficiency [%]				
		Total	Manifolds	Module		
				Total	FW	SB
#1	4.1	98.5	100	96.8	95.8	98.9
#2	4.1	98.5	100	96.8	95.8	99
#6-7	4.1	96.9	99.3	-	-	-
#6	-	-	-	92.9	91.4	96.3
#7	-	-	-	97.9	98.8	97
#6-7	2.1	96.8	99.5	-	-	-
#6	-	-	-	92	90.7	94.9
#7	-	-	-	98.3	99.1	97.5
#10S	4.1	98.1	97.5	99	99.5	98.3
#12C	2.1	97.2	96.4	98.8	99.3	98.1
#12S	2.1	96.8	95.8	98.8	99.3	98.1
#15S	4.1	98.9	99.4	97.6	96.6	99
#16C	2.1	97.7	99.8	95.2	99.3	91.9
#17C	4.1	100	100	100	100	99.9
#18S	4.1	99.9	100	99.7	100	99.4

Table 4. CD scenario: draining efficiency.

Cooling circuit	p_{in} [MPa]	Draining efficiency [%]				
		Total	Manifolds	Module		
				Total	FW	SB
#1	4.1	98.5	100	96.8	95.9	99
#2	4.1	98.6	100	96.9	95.9	99
#10S	4.1	98.1	97.5	99	99.5	98.3
#12C	2.1	97.2	96.4	98.8	99.3	98.2
#12S	2.1	96.8	95.8	98.8	99.3	98.2
#15S	4.1	99.3	99.4	98.9	98.5	99.4
#16C	2.1	97.7	99.8	95.4	98.8	92.7
#17C	4.1	100	100	100	100	100
#18S	4.1	99.9	100	99.7	100	99.4

From the results obtained it can be concluded that the cooling circuits considered in the analysis are generally able to be safely and quickly drained according to the envisaged forced draining procedure, with nitrogen injection pressures down to 2.1 MPa. Their comparative analysis indicate that draining transient duration lasts typically for ~40÷50 s under the reference scenario, increasing up to ~80÷90 s when the complete scenario is considered instead. Moreover, it may be observed that the residual water amount inside each cooling circuit varies generally from few tens of gram up to ~8 kilograms, allowing final draining efficiencies generally lower than 10% to be estimated. Residual water is typically located in the module cooling system and, particularly, in dead-end holes or as slugs trapped in meta-stable positions due to either an intense nitrogen counter-current flow or a horizontally stratified flow pattern.

As an example, Fig. 1 shows the trend of residual mass during the RD scenario with reduced injection pressure (RIP) of cooling circuit for twinned modules #6-7 and the residual water distribution is shown in Fig. 2, where volumes with more than 1 g of water are indicated in blue.

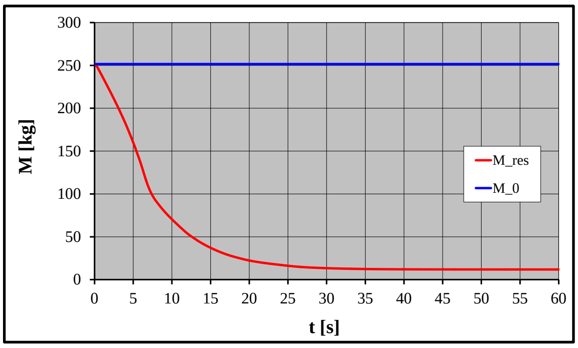

Fig.1. Reference draining scenario-RIP: residual mass vs. time.

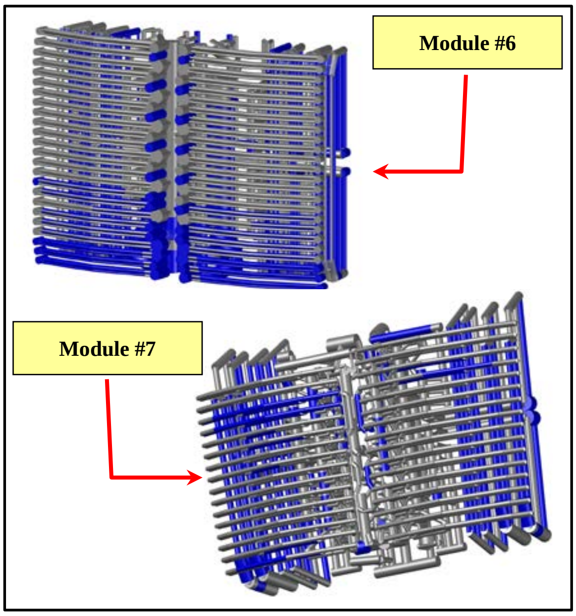

Fig.2. RD-RIP scenario M#6-7: residual water distribution.

4. Conclusions

A theoretical-computational analysis campaign has been performed by DEIM and IO to investigate the thermal-hydraulic behaviour of the ITER blanket standard sector cooling system under the D&D transient operational procedure, focussing the attention on the forced draining transient procedure relevant to the most critical and demanding modules of this blanket sector.

To this purpose, proper transient thermal-hydraulic analyses have been carried out with the RELAP5 system code, to assess the aptitude of each considered cooling circuit to be completely and quickly drained according to the envisaged operational procedure. The draining performances have been evaluated mainly in terms of transient duration, residual water distribution and draining efficiency.

From the results obtained it can be concluded that the cooling circuits considered in the analysis are generally able to be effectively drained according to the envisaged operational procedure, even reducing the nitrogen injection pressures down to 2.1 MPa. They indicate that the transient duration ranges from ~40÷50 s, under the reference scenario, up to ~80÷90 s, when the complete scenario is considered instead and inlet/outlet valves inertia is taken into account; the results are thus quite acceptable in all cases considered. Finally, residual water, mainly located inside the module cooling circuit within dead-end holes or as slugs trapped in meta-stable positions, is predicted to range from few tens of gram up to few kilograms, resulting in acceptable draining efficiencies lower than 10%.

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] P.A. Di Maio et alii, Analysis of the steady state thermal-hydraulic behaviour of the ITER blanket cooling system, this conference.

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

[8] R. E. Henry, H. K. Fauske, The Two-Phase Critical Flow of One-Component Mixtures in Nozzles, Orifices, and Short Tubes, Journal of Heat Transfer. 93 (1971).