Draining and Drying Process Development of the Tokamak Cooling Water System of ITER

Seokho Kim,[a] Walter Van Hove,[a] Juan Ferrada,[a] Pietro Alessandro Di Maio,[b] David Felde,[c] Mitteau Raphael, [d] Giovanni Dell'Orco,[d] Jan Berry [a]

[a] *US ITER, Oak Ridge National Laboratory, Oak Ridge, TN USA*
[b] *University of Palermo, Viale delle Scienze, 90128 Palermo, Italy*
[c] *Reactor and Nuclear Systems Division, ORNL, Oak Ridge, TN USA*
[d] *ITER Organization, 13067 St Paul Lez Durance, France*

The ITER Organization (IO) developed a thermal-hydraulic (TH) model of the complex first wall and blanket (FW/BLK) cooling channels to determine gas flow rate and pressure required to effectively blow out the water in the FW/BLK. In addition, US ITER conducted experiments for selected geometries of FW/BLK flow channels to predict the blowout parameters. The analysis indicates that as low as 2 MPa of pressure difference over the blanket modules will ensure substantial evacuation of the water in blankets with just a few percent remaining in the blanket flow channels. Validation of the thermal-hydraulic model and analysis against experimental data show a reasonable agreement between the modelling and experiment data and thus provide assurance of the modelling and analysis.

Keywords: First Wall, Blanket, Draining Process, Drying Process

1. Introduction

The ITER facility is an international research project that is constructed in Cadarache, France. The main objective of ITER is to demonstrate the scientific and technical feasibility of a controlled fusion reaction allowing the production of 500 MW of fusion power for durations of several hundred seconds. US ITER is responsible for the design, engineering, and procurement of the Tokamak Cooling Water System (TCWS). The TCWS transfers heat generated in the Tokamak to cooling water during nominal pulsed operation—850 MW at up to 126°C and 4 MPa water pressure. In addition to the heat removal function, TCWS provides baking for plasma facing components (PFCs), control of cooling water chemistry, and draining and drying for maintenance, and detection of a leaking PFC during an accident condition.

The current process of water removal from PFCs includes blowout with high pressure nitrogen gas, heatup of residual water in the PFC structure by hot nitrogen gas, evaporation/dryout of residual water, and then cooldown of the system to return to maintenance temperature. Development has focused on a blowout process since the other processes are relatively straightforward. A high level overview of the effort to develop the blowout process and the effort to validate theoretical models against experiment data are discussed in this paper.

2. Description of First Wall/Blanket System

One of the PFCs is the FW/BLK system which provides a physical boundary for the plasma transients and contributes to thermal and nuclear shielding of the vacuum vessel (VV) and external components. It consists of two major components: the shield block (SB) and the FW. The FW faces the plasma and is attached to the SB. The SBs are attached to the VV. Cooling water to the FW/BLK is supplied by manifolds supported off the VV behind the SB. The cooling water is fed to and from the SB through a coaxial connector and branch pipes. The FW/BLK system has both water supply and return lines connected to the cooling system at its highest elevation; hence, it is not possible to drain its water by gravity. Therefore, forced draining using pressurized nitrogen is a proposed operation to remove the water from the FW/BLK system.

3. TCWS Drying System Design

The TCWS is provided with a drying system (DYS) to dry out the in-vessel components using nitrogen gas in preparation for periodic leak testing or component replacement. The drying operation is performed after the system is gravity drained with residual water still trapped in the system; it involves a four-step process in the sequence given below:

Blowout: Pressurized nitrogen gas is blown through the system to remove as much of the water remaining at system low points as possible.

Heat-up: Hot nitrogen gas is circulated through the system to heat up in-vessel components. This prepares the system for evaporative removal of the water still remaining in the system.

Dry-out: Hot nitrogen gas (at reduced pressure if necessary) is circulated through the system for an

This manuscript has been authored by UT-Battelle, LLC under Contract No. DE-AC05-00OR22725 with the U.S. Department of Energy.

extended time to remove residual water by evaporation. The water vapor is condensed out and sent to drain tanks.

Cool-down: After the completion of the drying operations, the system is cooled down to the maintenance temperature.

Major DYS components include a compressor to circulate nitrogen gas through the system, an electric heater to heat up the gas, an economizer to conserve process heat, two condensers to cool and dry out the nitrogen gas, and a cyclone separator in series with a demister to separate and remove the water from the gas stream. The separated water is directed to drain tanks.

This paper focusses on the blowout phase. For the blowout to work, two conditions must be fulfilled. First, the pressure difference between inlet and outlet of the FW/BLK module must be sufficient to overcome the hydrostatic head down to the lowest point of the flow channel. Second, after clearing the FW/BLK module, the gas flow must be sufficiently high to entrain all the liquid water vertically upward and out of the cooling system. The verification of the latter condition is the subject of this paper.

4. FW/BLK Cooling Channel Draining Model

A thermal-hydraulic study was conducted to investigate performances of flow channels of a standard 20° blanket sector during the draining process. RELAP5 Mod3.3 was used to model blankets and their flow channels [1]. A similar study was performed earlier to study thermal-hydraulic behaviour of the divertor cassette, the upper port plug trapezoid section, and the test blanket module port plug, both under steady state and draining transient conditions [2–4].

The study was focused on flow channels of the most critical and demanding modules of a standard blanket sector to assess blowout performance to the operational procedure. For this purpose, the residual amount of water inside flow channels was calculated as well as its spatial distribution. Specific attention was paid to the potential effect of nitrogen injection pressure variation on the cooling system draining performances.

4.1. Thermal-Hydraulic Model

A finite volume model was developed for each of the blanket module flow channels. The geometrical model realistically reproduces in a quasi-2D way the flow domain, including inlet/outlet manifolds up to the upper ring manifold subheaders. Upstream and downstream conditions were modeled by fixed-pressure boundaries. The constitutive models provided by the RELAP5 code were adopted to describe the thermodynamic behaviour of water and/or nitrogen. The ideal gas model was assumed for nitrogen gas. The hydraulic model simulates single-phase water flow and two-phase water-nitrogen flow inside the flow channels. In particular, concentrated and distributed hydraulic resistances were modeled as a function of flow velocity. The Henry-Fauske model was used to model a choked flow condition [5]. Details about the models as well as on their predictions under nominal steady state conditions can be found in [6].

4.2. Blowout process analyses

A realistic scenario for the blowout transient was considered. The study assumed that compressed nitrogen of 4.1 MPa and 50°C was blown into blanket flow channel that was initially full of water at 50°C and 0.1 MPa. A parametric study was also conducted for nitrogen injection pressure down to 0.6 MPa to investigate those critical flow channels where draining behaviour might be considered poor due to their position and/or configuration.

Main results were published in detail elsewhere [7]. A conclusion was that the considered flow channels could generally be quickly cleared with compressed nitrogen of as low as 2.1 MPa. In particular, the analysis indicated that the blowout process typically lasted less than 1 min, increasing up to 80~90 s when inlet/outlet valves with their resistances are considered. Moreover, results showed that the amount of residual water ranges from a few tens of grams up to a few kilograms, resulting in blowout efficiencies higher than 90%. Residual water was typically located 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 depicted in Fig. 1. In this figure, residual water >1 g is indicated in dark colour for module 1 flow channel, with 4.1 MPa nitrogen blowout pressure with inlet/outlet valves on blanket manifold piping.

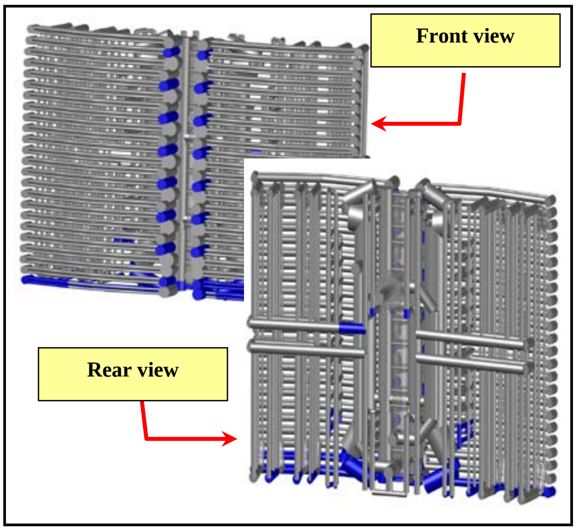


Fig.1. Residual water distribution in module-1.

A parametric study highlighted the fact that the amount of residual water decreased as nitrogen blowout pressure increased (Table 1), showing a sharp reduction in the interval 1.1~2.1 MPa of pressure drop. No significant residual water variations were shown for nitrogen blowout pressures higher than 2.1 MPa. Results also indicated that nitrogen blowout pressure higher than 1 MPa is recommended for modules 6-7, which are connected in parallel to the manifold pipes (Fig. 2).

Table 1. Amount of residual water (kg).

Blanket Number	Nitrogen pressure drop [MPa]							
	0.5	0.8	1	1.25	1.5	2	3	4
1	6.3	-	6.56	4.95	3.42	3.23	-	2.6
6–7	-	13.5	9.38	-	8.67	8.11	-	7.81
10S	7.3	-	6.01	-	4.94	4.07	3.58	3.39
18S	1.1	-	0.72	0.35	0.30	0.23	-	0.16

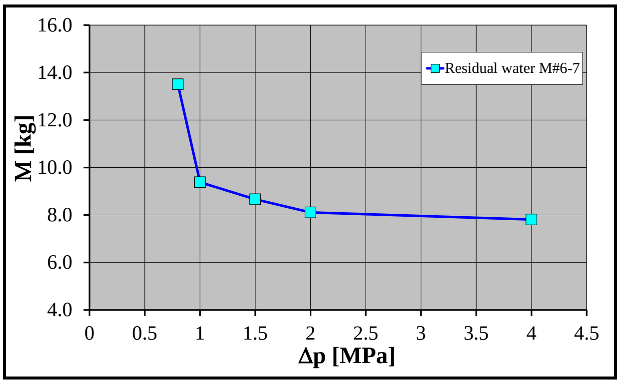
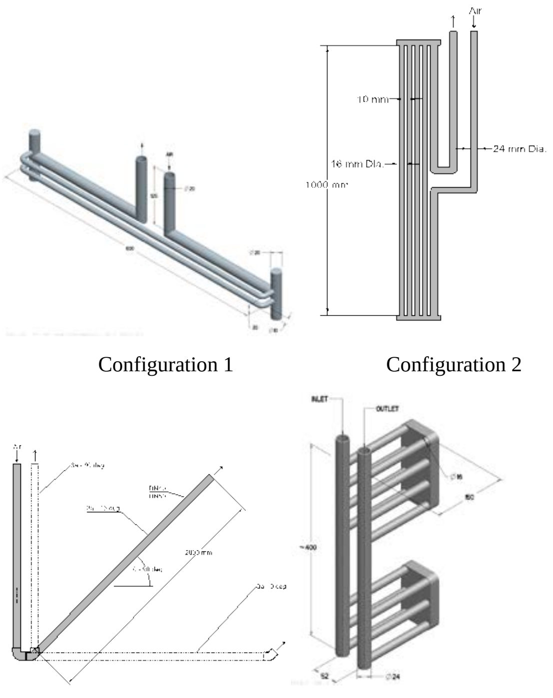


Fig. 2. Residual water versus pressure drop for modules 6 & 7.

5. Blowout Experiments

A few representative flow channels of the FW/BLK module and manifold geometries were tested in order to evaluate the effectiveness of using pressurized nitrogen to remove water from the system. Countercurrent flow between water and nitrogen and possible stratification of water in horizontal legs were of particular interest for the design. Four configurations were fabricated and tested, along with orientation variations for some of the configurations shown in Fig. 3. Most of the tests were conducted at low pressure with the air/water mixture exhausted to a tank at atmospheric pressure. The low pressure allowed the use of clear PVC piping and cast acrylic in the fabrication of piping and manifold sections, providing a view of the air/water interaction through these transparent materials. High pressure tests were also performed for the configuration 4 geometry using a stainless steel assembly to examine the effects at more prototypic pressures and to provide a comparison between results with cast acrylic material and 316 stainless steel. A simplified sketch with a brief description indicating the portion of the ITER blanket modules represented and the objective for testing is shown in Fig. 3 for each of the configurations.

A schematic of the low-pressure test setup is shown in Fig. 4. Compressed air is supplied via a connection to the 1/4-inch building instrument air supply line (nominally at 0.7 MPa and ambient temperature). The inlet air line includes a Coriolis mass flow meter, a pressure transmitter, and pressure gauges to monitor conditions in the line and at the inlet of the test assembly. With a ball valve, HV-1, a manual V-notch ball valve, HCV-2, controls airflow to the test assembly. The air-water mixture in the outlet line from the test assembly flows to a vented tank via a larger 102 mm (4 in.) PVC pipe where most of the water drops out of the flow stream. The tank is installed on a scale with output transmitted to the data acquisition system to provide a measurement of water removed from the test section as a function of time. Air exits the separator tank via a 102 mm (4 in.) PVC pipe. The transparent materials used in fabricating the assemblies for the low pressure testing allowed visual observation of the internal flow channels and manifolds. An Olympus i-SPEED2 high-speed video camera was used to capture movies and still images for some of the tests.



Configuration 1 Configuration 2

Configuration 3 Configuration 4

Configuration 1: Cooling channels in the FW "fingers" (Blowout horizontal stratified flow)
Configuration 2: Shield Block internal cooling pipes (Blowout parallel channels)
Configuration 3: Manifold piping and distribution piping (Counter-Current Flow Limitation - CCFL in manifolds and distribution pipes)
Configuration 4: Shield Block internal cooling pipes of various orientations (SB cooling pipes for observing behavior of parallel pipes and water hold-up in headers)

Fig. 3. Test configuration designs for the blowout tests.

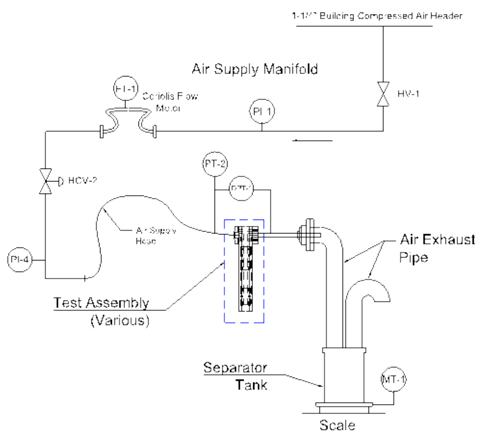


Fig. 4. Low-pressure system.

The general test method involved setting the V-notch ball valve, HCV-2, to the desired air flow rate with the system empty, closing an upstream ball valve, HV-1, on the air supply line, and then filling the system with water. The test was initiated by fully opening HV-1 (over a few second intervals) with airflow rate increasing to the condition set by the HCV-2 position. Airflow was generally maintained for 30–40 s. This time period was generally sufficient to establish more or less steady airflow rates through the assembly and remove the bulk of the water for the flow condition. Leaving the airflow on longer would slowly remove more water by agitation and evaporation, but for comparison purposes the 30-40 s duration seemed appropriate.

Measurement of water remaining in a test assembly for a given flow condition was made in different ways depending on the test assembly geometry and size. Although the transient scale reading of water collected during the test is available (MT-1), using this measurement was generally not the best method for determining water remaining in the test assembly due to holdup of water in the exhaust piping and potential evaporation and carryover of water in the exhaust. In some cases, it was practical to disconnect the test assembly from the system and weigh the assembly before and after a flow test. For larger assemblies, visual measurements of water heights or comparisons of camera images of known water volume were used.

Some limited results for configurations 2 and 3-45deg are presented for comparison with the analytical modeling results. Some typical results are shown for tests with the configuration 2 assembly in Fig. 5. Water remaining is plotted for a range of air mass flow rates. Essentially all of the water is removed at 70 kg/h air flow, with only a few small beads remaining in the bottom of the lower manifold section (less than 1–2 ml) based on visual observation.

Table 2. Gas mass flow rate required removing most water.

Configuration	Estimate of required gas flow rate (kg/h)	Initial fill (g)	Estimate of residual water (g)
1	25	330	< 5
2	70	2000	< 5
3-0 deg	200	4768	< 5
3-45 deg	200	7433	< 5
3-90 deg	200	8577	<10
4-Vertical	95	1230-1240	< 15
4-Horizontal	95	1449-1559	< 15
4-52 deg	140	1235-1241	< 10
4-Vertical (2 MPa)	435	1240	< 15
4-52 deg (2 MPa)	450-500*	1240	< 50

*Extrapolated value - Gas mass flow rate available limited by downstream flow control valve to ~ 435 kg/h for 2 MPa test pressure.

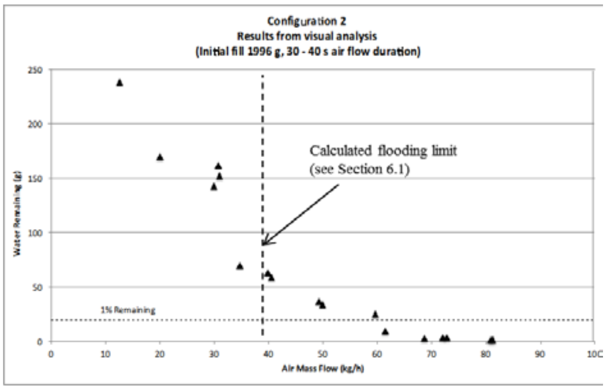


Fig. 5. Residual water mass as a function of air mass flow for configuration 2.

The inlet pressure, the air mass flow, and the scale reading were recorded during the tests. An example of the transient data recorded for a configuration 3-45deg test is shown in Fig. 6. Most of the water is removed relatively early in the transient with the scale reading flattening out after ~15 s. The drop in scale reading near the end is the effect of the air momentum into the separator tank being removed after closing the air supply valve.

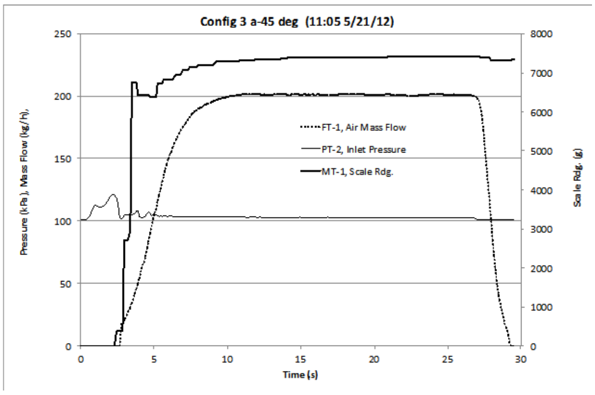


Fig. 6. Example of transient data for test to 200 kg/h for configuration 3-45 deg (initial fill of 7433 g).

A summary of the 'water remaining' data for the various configurations is provided in Table 2. The column "Estimate of required gas flow rate" is the best estimate of the flow required to remove most of the water from the system and is based on the plots of "Water Remaining vs. Gas Mass Flow Rate." Except for the 2 MPa tests, water removal of more than 99% was achievable for all cases. For the 2 MPa case, and particularly the 52 degree orientation, the downstream flow control valve limited the flow rate so that the estimate required extrapolation to the mostly empty condition.

The duration of the gas flow is in the range of 30–40 s for most of the tests, in order to focus primarily on the dynamic effects of the applied gas flow rate. Water remaining for most of these cases was in the range of a few milliliters to a few tens of milliliters. Generally, further water removal was obtained and remaining small beads were removed at higher flow rates (increasing by

20–30%) and extended flow duration times of a few more minutes.

6. Blowout Model Verification

6.1 Scaling and Analytical Validation

One of the major conditions needed for the blowout process to work is that the upward flowing gas must be able to entrain the liquid out of the system. This requires that the interfacial friction between liquid and gas can overcome the gravity force on the liquid such that the gas can entrain all the liquid upwards. This is known as counter-current flow limitation (CCFL). The counter-current flow of a liquid and a gas is traditionally described by the Wallis correlation [8]:

$$J^{*1/2}_g + m\,J^{*1/2}_l = C \qquad (1)$$

where J^*_g and J^*_l are, respectively, the nondimensional upward gas and downward liquid velocities. Constants "m" and "C" in Eq. (1) depend on the channel exit conditions. The minimum gas velocity required to prevent any liquid from flowing downwards, counter-current to the gas flow, is called the flooding limit and is given by:

$$J^{*1/2}_g = C \qquad (2)$$

In most of applications, a recommended value for C is on the order of C ≈ 1. Assuming that liquid density is much larger than gas density, $\rho_l >> \rho_g$, which is the case in the experiments reported here. Eq. (2) can be transformed in an equation giving the minimum required gas flow:

$$w_g = (\rho_g \rho_l g D)^{1/2} A C^2 \qquad (3)$$

where ρ_l is the liquid density; ρ_g is the gas density; g is the acceleration of gravity; D is the hydraulic diameter; A is the flow area; and C is the defined constant above.

Applying Eq. (3) to configuration 2, the required airflow is 27 kg/h for the single return pipe and 40 kg/h for the four parallel pipes. The upward flow through the four parallel pipes is limiting in this case, and the calculated airflow is indicated by the vertical dotted line in Fig. 5. The results of the experiments for configuration 2 showed the existence of a threshold value of around 35 kg/h. Below this airflow, not all four parallel pipes are cleared. The experiments indicate that a minimum of 35 kg/h of airflow is needed to clear all four parallel pipes. The threshold is clearly visible in Fig. 5, and the Wallis correlation predicts a value above this threshold.

Configuration 3 is constructed of DN50 Sch40 pipe. The above equation for the required minimum airflow gives 190 kg/h in this case. This value is compared with the experimental values of the remaining water for the vertical configuration 3-90 in Fig. 7. The calculated flooding limit is indicated by the vertical dotted line. The Wallis correlation gives a good prediction of the required minimum airflow in this case.

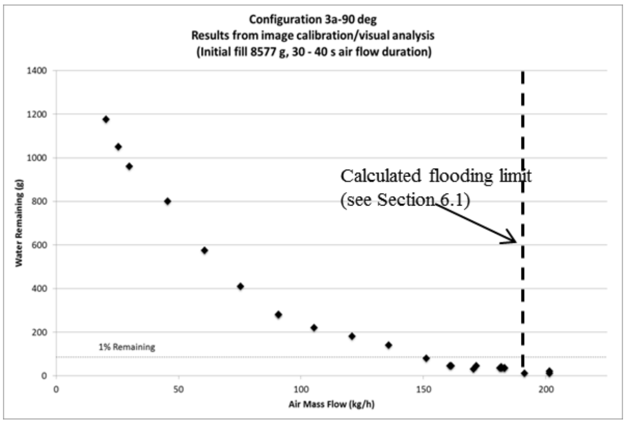


Fig. 7. Water remaining as a function of air mass flow for configuration 3-90.

The Wallis correlation is a nondimensional equation that allows scaling the results to other operating conditions. Because in reality the blowout will be performed at high pressure, we are most interested in the effect of pressure on the CCFL. A limited number of experiments were performed for the same geometry but at higher gas pressure (see Table 2).

Assuming the same geometry and cold water as liquid with the same density, only the gas density in the above equation changes between the low and high pressure experiments. Furthermore both series of experiments were performed at ambient temperature. For convenience, the high pressure experiments were performed with pressurized nitrogen. But the difference in the specific gas constants for air and nitrogen can be neglected. The scaling law between the low and high pressure experiments then becomes very simple:

$$\frac{w_{g1}}{w_{g2}} = \left(\frac{p_1}{p_2}\right)^{1/2} \qquad (4)$$

where p is the pressure of the experiment and the indices 1 and 2 refer to the high and low pressure experiment, respectively, on the same geometry. This scaling law is incidentally also the scaling law for the gas flow giving the same pressure losses through the same geometry but at different pressures. This relation implies that in a 2 MPa experiment, the minimum flow to remove all the water should be 4.5 times higher than in a 0.1 MPa experiment. This is confirmed by the results in Table 2. The configuration 4-Vertical needs 95 kg/h at 0.1 MPa to bring the residual water down below 15 g. According to Eq. (4), 430 kg/h would be needed at 2 MPa. The experimental value was 435 kg/h.

The experiments confirm that the Wallis correlation with C=1 can be used to calculate the minimum required gas flow for blowout.

6.2 Validation of BLK RELAP Model against Selected Test Results

To validate a RELAP model used for the blanket blowout analysis, simple RELAP models were developed for each configuration of the experiments described above. The models used the same approach as those used in the blanket flow channel thermal-hydraulic study. In this paper, the result is described below for configuration 2. The inlet pipe was modeled as directly connected to an "infinite" volume of air at 20°C. The outlet manifold was modeled to discharge to a tank at atmospheric pressure, which was simulated as an "infinite" time-dependent volume. Water/air was initially at atmospheric pressure and 20°C. The smooth pipe approximation was used for the test assembly with negligible small wall roughness.

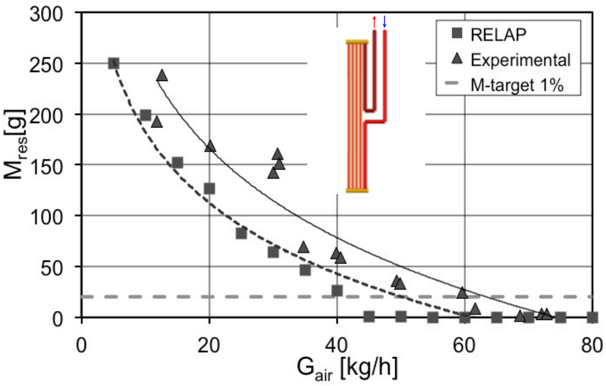


Fig.8. Residual water mass from the RELAP analysis and the experiment for configuration 2.

Both experimental and calculated residual water mass are shown in Fig. 8. The residual water mass was calculated to be less than the measured value for the same air mass flow. For complete blowout, the RELAP analysis results in ~60 kg/h of airflow, while the experiment yields ~73 kg/s. This agreement is sufficient to validate the model, given very rapid fluid dynamics occurring in the flow channels. However, the validation could be further improved by modelling accurate boundary conditions, such as the inlet pressure time dependency. Validation to configuration 4 (not shown here) also showed a similar agreement. These validations give an acceptable confidence in the RELAP model and analysis described above.

7. Summary and Conclusions

To determine design parameters for the nitrogen blowout flow rate and pressure, a thermal-hydraulic model was developed for the FW/BLK modules using a RELAP code. The analysis indicates that as low as 2 MPa of pressure difference over the blanket modules will ensure substantial evacuation of the water in blankets with just a few percent remaining in the blanket flow channels. For modules 6-7, which are connected in parallel to the manifold pipes, results also indicate lower blowout efficiency than a single module case.

To validate the RELAP model that was used for the blowout analysis, a few representative flow channels of the FW/BLK module and manifold geometries were tested with special interest in CCFL between water and gas and possible stratification of water in flow channels. All the tests showed substantial removal of water in test flow channels in a very short period of time. CCFL condition was analyzed using the Wallis correlation for configurations 2 and 3-90 deg. The calculated flooding limit is slightly lower than the experimental data for configuration 2; however, both the analytical and experimental flooding limit matches very well for configuration 3-90deg.

A high pressure blowout test was conducted with 2 MPa airflow for configuration 4 to qualify all other tests which were conducted with low blowout pressure. This scaling exercise results in 4.5 times more flow needed to blowout for 2 MPa pressure than for low pressure cases. For configuration 4-Vertical at low pressure, 95 kg/h is needed to blow out (from table 2) while the same configuration for a high pressure requires 435 kg/h, which is 4.6 times higher flow for a high pressure blowout. Experimental observation matches very well with a theoretical scaling so that we can imply findings from low pressure tests to real design of high pressure with a confidence.

To validate a RELAP model used for the blanket blowout analysis, simple RELAP models were developed for each configuration of the experiments. Results for configurations 2 and 4 indicate that predicted residual mass agrees reasonably well with experimental data, and thus this validation provides assurance of the modelling and analysis.

References

[1] RELAP5 Mod3.3 code - Theory Manual, INEL Nuclear Safety Analysis Division.
[2] A. Tincani, et al, Steady state and transient thermal-hydraulic characterization of full-scale ITER divertor plasma facing components, Fusion Engineering and Design 83 (2008) 1034-1037.
[3] P. A. Di Maio, et al, 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.
[4] B. Y. Kim, et al, Status of ITER TBM port plug conceptual design and analyses, Fusion Engineering and Design 89 (2014) 1969-1974.
[5] 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).
[6] P. A. Di Maio, et al, Analysis of the steady state thermal-hydraulic behaviour of the ITER blanket cooling system, Fusion Engineering and Design, in press, 10.1016/j.fusengdes.2015.05.070.
[7] 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 Engineering and Design, in press, 10.1016/j.fusengdes.2015.01.024.
[8] G. B. Wallis, One-Dimensional Two-Phase Flow, McGraw-Hill (1969).