G Model
FUSION-8684; No. of Pages 7

ARTICLE IN PRESS

Fusion Engineering and Design xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Fusion Engineering and Design

journal homepage: www.elsevier.com/locate/fusengdes

On the hydraulic behaviour of ITER Shield Blocks #14 and #08. Computational analysis and comparison with experimental tests

P.A. Di Maio [a], M. Merola [b], R. Mitteau [b], R. Raffray [b], E. Vallone [a,*]

[a] Dipartimento di Energia, Ingegneria dell'Informazione e Modelli Matematici, Università di Palermo Viale delle Scienze, 90128, Palermo, Italy
[b] ITER Organization, Route de Vinon sur Verdon, 13067 Saint Paul, Lez Durance, France

HIGHLIGHTS

- A benchmarking activity has been carried out focusing the attention on the cooling circuits of ITER Shield Blocks #08 and #14.
- A theoretical-computational fluid-dynamic approach based on the Finite Volume Method has been followed, adopting a commercial code.
- Hydraulic characteristic functions and spatial distributions of coolant mass flow rate, velocity and pressure drop have been assessed.
- Results obtained have allowed code benchmarking for Blanket modules and the numerical predictions have been found to be generally lower than but quite close to the experimental results (lower than 10%).

ARTICLE INFO

Article history:
Received 31 August 2015
Received in revised form 8 March 2016
Accepted 14 March 2016
Available online xxx

Keywords:
Blanket
CFD analysis
Hydraulics

ABSTRACT

As a consequence of its position and functions, the ITER blanket system will be subjected to significant heat loads under nominal reference conditions. Therefore, the design of its cooling system is particularly demanding. Coolant water is distributed individually to the 440 blanket modules (BMs) through manifold piping, which makes it a highly parallelized system. The mass flow rate distribution is finely tuned to meet all operation constraints: adequate margin to burn out in the plasma facing components, even distribution of water flow among the so-called plasma-facing "fingers" of the Blanket First Wall panels, high enough water flow rate to avoid excessive water temperature in the outlet pipes, maximum allowable water velocity lower than 7 m/s in manifold pipes. Furthermore the overall pressure drop and flow rate in each BM shall be within the fixed specified design limit to avoid an unduly unbalance of cooling among the 440 modules.

Analyses have to be carried out following a computational fluid-dynamic (CFD) approach based on the finite volume method and adopting a CFD commercial code to assess the thermal-hydraulic behaviour of each single circuit of the ITER blanket cooling system.

This paper describes the code benchmarking needed to determine the best method to get reliable and timely results. Since experimental tests are available in ITER Organization on full scale prototypes of Shield Blocks #08 and #14, CFD analyses have been performed to investigate their fluid-dynamic behaviour under steady state conditions and compare the numerical and experimental results. Results obtained are presented and critically discussed.

1. Introduction

The blanket system represents one of the pivotal components of the ITER reactor (Nuclear Facility INB-174), providing a physical boundary to the plasma 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 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 Coils and

* Corresponding author.
E-mail address: eug.vallone@gmail.com (E. Vallone).

http://dx.doi.org/10.1016/j.fusengdes.2016.03.060
0920-3796/

Please cite this article in press as: 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. (2016), http://dx.doi.org/10.1016/j.fusengdes.2016.03.060

Divertor (IBED) Primary Heat Transfer System (PHTS) of the ITER Tokamak Cooling Water System (TCWS) [2].

As a consequence of its position and functions, the blanket system will be subjected to significant heat loads under nominal reference conditions, and the design of its cooling system is particularly demanding. In fact, it has to ensure that adequate cooling is provided to each module to promote convective heat transfer and to prevent burn out in its plasma facing components while complying with ITER pressure drop and flow velocity limits to avoid an unacceptably high pumping power and minimise corrosion concerns.

The Department of Energy, Information Engineering and Mathematical Models (DEIM) of the University of Palermo has collaborated with the ITER Organization (IO) on the assessment of the ITER blanket cooling system hydraulic performances under both steady state and draining and drying transient operational conditions [2,3].

A theoretical-computational research campaign has been launched following a computational fluid-dynamic (CFD) approach based on the finite volume method and adopting a qualified CFD commercial code to assess the steady state thermal-hydraulic behaviour of selected cooling circuits of the ITER blanket cooling system.

In particular, a code benchmarking activity has been carried out to determine the best method to get reliable and timely results, focusing the attention on the numerical assessment of the steady state hydraulic performances of the cooling circuits of Shield Blocks #08 and #14, for which experimental results of full scale prototype tested by the Korean and Chinese ITER Domestic Agencies, are available. This would then allow a direct comparison between numerical and experimental results.

This paper summarizes the code benchmarking activity, describing the discretization and modelling strategies adopted and critically comparing the numerical and experimental results obtained for the SBs #08 and #14 cooling circuits steady state fluid-dynamic behaviour.

2. ITER blanket cooling system

The ITER blanket cooling system protects the blanket structure against overheating and extracts the nuclear deposited heat power. It relies on the use of sub-cooled water at inlet temperature and pressure of 70 °C and 4.0 MPa, respectively, and is mainly composed of the following components [1]:

- SBs cooling circuits;
- FWs panels cooling circuits;
- hydraulic connectors;
- blanket manifolds system.

These components are arranged in 363 different cooling circuits, connected in parallel up to the TCWS Upper Ring Manifold and providing cooling water to the 440 separate modules of the ITER blanket. Most circuits individually cool a single blanket module, while some cool two modules in parallel and a few three modules in parallel, in order to comply with space constraints [4].

Each circuit is fed with a proper mass flow rate of water coolant (typically ranging from 4.7 kg/s to 9.6 kg/s for each blanket module), that enters from the inlet manifold being first routed to the inlet flexible pipe and, then, to the FW cooling circuit, from which it passes, through the outlet flexible pipe, to the SB cooling circuit. After circulating through the SB cooling channels, the coolant is finally routed into the outlet manifold, from which it is delivered back to the TCWS Upper Ring Manifold.

3. Hydraulic research campaign

As consequence of its architecture, the blanket cooling system is a highly parallelized network of 363 circuits, whose mass flow rates have to be properly tuned in order to guarantee the safe extraction of the 736 MW of thermal power under reference nominal conditions, while complying with operation constraints and requirements.

Therefore, a theoretical-computational research campaign has been launched by IO to assess coolant mass flow rate distribution inside each blanket cooling circuit so to check whether a well-balanced cooling would be ensured among the modules, fulfilling IO cooling system requirements in term of pressure drop, temperature increase, flow velocity and margin against critical heat flux occurrence.

A CFD approach based on the finite volume method has been adopted to carry out the research campaign and a suitable release of the ANSYS-CFX CFD code [1] has been selected as the reference computational tool. It is a well-known and qualified CFD commercial code intended to simulate in a fully 3D approach the thermofluid-dynamic behaviour of fluid flows. In particular, it numerically solves the fluid flow equations by adopting an element-based finite volume method, particularly suitable for analysing the thermofluid-dynamic behaviour of single-phase, single-component flows both in laminar and fully turbulent regimes.

A preliminary research activity has been carried out in close cooperation between DEIM and IO to assess clear guidelines for the CFD analyses in order to obtain reliable computational results within viable calculation times as well as to check the predictive potential of the ANSYS-CFX code for fusion-relevant applications. In particular, attention has been focussed on the CFD analysis of the steady state hydraulic behaviour of Shield Blocks #08 and #14, whose full-scale prototypes (FSP) have been already experimentally tested allowing for a reliable benchmarking of the ANSYS-CFX code.

3.1. Experimental tests

FSP of SBs #08 and #14 (Fig. 1) have been commissioned by IO and manufactured by the Korean and Chinese ITER Domestic Agencies, respectively. Thereafter, they have undergone experimental tests to investigate their steady-state hydraulic behaviour by assessing their hydraulic characteristic functions, $\Delta p = \Delta p(G)$, and in particular the pressure drop across the component, Δp, as a function of the corresponding mass flow rate, G, under steady state conditions [6].

3.2. Numerical analyses

A set of CFD parametric analyses has been performed for each SB cooling circuit investigated, to assess the pressure drop, Δp_i, needed to let each tested mass flow rate, G_i, flow through the circuit. The $(G_i, \Delta p_i)$ pairs numerically assessed have been best fitted with a power-law analytical form and they have been compared to the experimental ones, allowing code benchmarking.

3.3. Finite volume models

Two 3D finite volume models have been properly set-up to realistically reproduce the geometric and flow features of SBs #08 and #14 flow domains (Figs. 2 and 3). Inlet and outlet sections have been chosen according to the technical specifications of DAs hydraulic tests.

Please cite this article in press as: 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. (2016), http://dx.doi.org/10.1016/j.fusengdes.2016.03.060