Numerical Flow Models for controlled Fusion, April 16-20, 2007, PORQUEROLLES, France, organized by INRIA

Thermal-hydraulics of Plasma Facing Components for controlled fusion

F. Escourbiac[a], S. Pascal-Ribot[b], P. Spitz[a], M. Grandotto[b]

[(a)] *Association Euratom-CEA, CEA/DSM/DRFC, CEA/Cadarache, F-13108 SAINT PAUL LEZ DURANCE, France*
[(b)] *, CEA/Cadarache, F-13108 SAINT PAUL LEZ DURANCE, France*

First generations of controlled fusion machines were dedicated to physics issues and characterized by short plasma pulses : a radiative cooling in-between shots was enough to control the temperature of the device. Nowadays; the emergence of steady-state controlled fusion machines imposes energy from the plasma has to be exhausted by a fluid.

Consequently, the whole surface of the vacuum chamber has to be covered by heat exchangers - so-called “Plasma Facing Components” (PFC) - cooled with pressurized water. In the most loaded parts, the PFC must handles heat flux in the range of 20 MW/m² at steady-state which is one order of magnitude more than in Pressurized Water Reactor. Furthermore, these elements are heated only on one side. These specific design requirements generate a number of engineering and operational constraints, amongst them the necessity to work in the subcooled boiling regime and to eventually use turbulence promoters like helical inserts into the tube, or particular geometries like the hypervapotron.

Experimental studies have been undertaken in the fusion community since the 1990s to determine correlations for onset of nucleate boiling, partial then fully developed subcooled boiling regime and critical heat flux, essentially depending on wall temperature, bulk fluid velocity, and empirical coefficients. The latest generation of PFC (Tore Supra upgrade, W7-X, ITER) was designed with these experimental results used as boundary conditions for thermal calculations. However, the use of empirical coefficients limits drastically the predictive possibility of this type of simulation, particularly for some design modifications such as the hypervapotron and other complex geometries.

More recently, a coupling between a 3D two-phase flow thermal-hydraulics code (the Computational Fluids Dynamics module of NEPTUNE Code) and a 3D solid thermal code was developed to better understand the flow inside hypervapotron geometry and, if possible, to envisage a predictive model for geometrical and thermo-hydraulic optimisation. The wall heat flux description was based on the Podovsky model, modified by further work. The closure laws for the 4 contributions were identified - single phase heat flux (vapour and liquid), evaporation flux and quenching flux. Coupling feasibility and code robustness were demonstrated. 3D simulations were performed and compared satisfactorily with experimental results studying the possibility of using the hypervapotron cooling concept for ITER application.

This paper will introduce the problem of plasma facing components, briefly review the subcooling regime and finally give recent results on computational fluids modelling of the hypervapotron cooling concept.