Thermal Issues in Emerging Technologies, ThETA 2, Cairo, Egypt, Dec 17-20th 2008

EVALUATION OF DIFFERENT COOLING FLUIDS FOR HIGH-VOLTAGE NEUTRAL BEAM INJECTOR GRIDS

Piero Agostinetti, Marco Boldrin and Francesca Fantini

Consorzio RFX, Associazione Euratom-ENEA sulla fusione, Padova, Italy

ABSTRACT

The Neutral Beam Injectors (NBIs) of the ITER experimental fusion reactor are designed to accelerate Deuterium negative ions with energy up to 1 MeV and current up to 40 A. The accelerator grids must be designed to operate at high voltages and to withstand high power densities (in the order of some tens of MW m^{-2}). They must maintain a proper alignment in all the foreseen operating scenarios, in order to obtain good beam optics, so the thermo-mechanical deformations must be maintained at very low values. Further requirements come from the need of keeping under control the maximum surface temperature in copper.

With these requirements, the cooling of the grids represents a significantly critical aspect of the NBI design. Coolant properties have to satisfy high resistivity requirements and to be appropriate for the removal of high heat loads. The cooling circuits must match with the beam optic geometry and the space constrains severely affect the coolant distribution.

This paper presents some studies of the grid cooling circuits design carried out with Computational Fluid Dynamics (CFD) numerical simulations and analytical methods. Cooling performances for different cooling fluids (water and dielectric coolants) have been investigated.

1. INTRODUCTION

The Neutral Beam Injectors (NBI) for the International Thermonuclear Experimental Reactor (ITER) have to supply 16.7 MW each of additional heating power to ignite the plasma, accelerating negative ions up to 1 MeV with a beam current up to 40 A [1].

The present work, carried out in the framework of the grids cooling system design activities, deals with the choice of a proper cooling fluid for some components of the NBI, that must be maintained to high potential (up to 1 MV) and are subjected to high heat fluxes (up to 20 MW m^{-2}). The reference solution with water cooling is compared with an alternative one using alternative dielectric cooling fluids, considering both the electrical and thermo-mechanical aspects. The advantages and drawbacks of each solution are analyzed and discussed.

2. HIGH VOLTAGE REQUIREMENTS AND EVALUATION OF DIFFERENT FLUIDS

A functional scheme of the NBI is presented in Figure 1. Inside the Neutral Beam injector vessel negative ions (Hydrogen in a first experimental phase and Deuterium for the final operating condition) are generated in the plasma source referenced at -1MV DC potential, and then extracted and accelerated from -1MV to ground by means of extractor and accelerator grids biased at increasing potentials.

The plasma source, extractor and accelerator (together referred as beam source) are fed by dedicated power supplies through a multi conductor SF_6 pressurized Transmission Line, whose highest potential electrode (-1MV) contains all the power and measurement conductors for the beam source components. In a pressurized tank, namely High Voltage Deck 2 (HVD2), are hosted the cooling pipes for the beam source together with the Hydrogen / Deuterium gas feeding line.

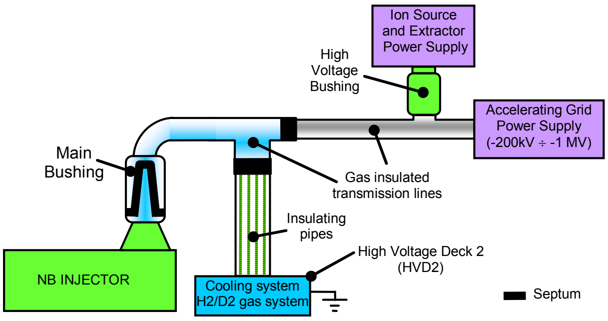

Figure 1. Functional representation of the NBI

To bring coolant fluids from ground potential to high voltage components, insulating pipes with suitable length are necessary to limit leakage currents due to fluid conductivity. Presently, for high voltage components the design foresees the use of ultrapure water (according to Type I ASTM D1193-91 standard characteristics). In this case, assuming that:

- ✓ water with 35 °C inlet temperature is used, characterized by a resistivity of $\rho \cong 11$ MΩ·cm;
- ✓ the pipes length is fixed at L = 4.4m

then the electrical power (P_u) dissipated per unit of water volume (U) when V = -1MV DC electric potential is applied would be:

$$P_u = \frac{V^2}{\rho \cdot L^2} = 470\frac{kW}{m^3} \equiv 470\frac{W}{l} \qquad (1)$$

As a matter of fact, during NBI operation the outlet water temperature becomes higher, with a remarkable conductivity increase (up to 10 times if 100°C are reached) as shown in Figure 2 [2]. The power losses increase at the same way.

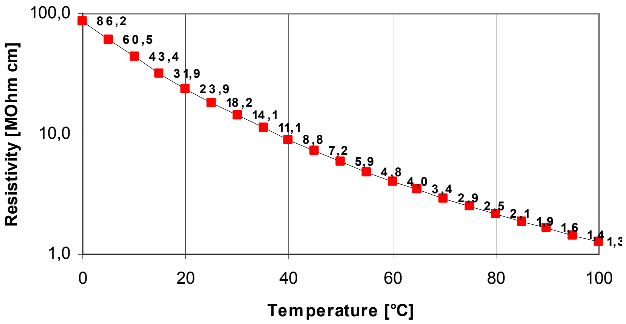

Figure 2. Ultrapure water Resistivity vs Temperature

Under these conditions, it would be necessary to increase the length of the pipes coming from HVD2 in order to maintain nearly the same power dissipation. This would lead to excessive HVD2 dimensions.

Such huge power dissipation in water has been never faced neither in industrial nor in research applications; in any case, the issue of controlling the water related processes like:

- possible localized water boiling due to joule power dissipation with production of vapour bubbles;
- electrolysis effects causing gas production (H_2) and electrode corrosion (production of Cu compounds like oxides [3]);

could become too difficult to be solved; this would turn in a reduced reliability and availability of the NBI system, not acceptable by the ITER Risk Assessment Management (RAM) criteria.

An attractive alternative to water could be the use of an insulating and radiation resistant fluid. For this purpose, it was considered worthwhile to investigate the effectiveness of the use of perfluorinated fluids. These fluids, colourless and odourless, have been used for over 40 years to cool electronic devices for military and computer applications. They offer excellent dielectric properties, good properties as heat transfer media and good materials compatibility. Furthermore they are thermally and chemically stable.

For NBI ITER application, a suitable choice could be Fluorinert™ FC77, which is characterized by a boiling point of 97°C [4].

Table 1 summarize main characteristic of this liquid with respect to water (both at room temperature). It can be observed that electrical resistivity is very high whilst the cooling capabilities are lower than water.

The main advantages would be:

- absence of electrochemical corrosion;
- very small leakage current;
- simplification of the cooling circuit (no need to build a very long water column);
- less demanding chemical control system;
- more reliability for 1 MV holding insulation.

Table 1. Main physical characteristics comparison

	Fluorinert FC-77 (@25°C) (a)	**Water (@25°C) (b)**	***Ratio (a) / (b)***
Density *$[kg\cdot m^{-3}]$*	1780	997	*1.79*
Specific heat Capacity *$[kJ\cdot kg^{-1}\cdot K^{-1}]$*	1.10	4.18	*0.26*
Thermal conductivity *$[W\cdot m^{-1}\cdot K^{-1}]$*	0.063	0.608	*0.10*
Kinematic viscosity *$[m^2\cdot s^{-1}]$*	$7.2\cdot10^{-7}$	$8.937\cdot10^{-7}$	*0.81*
Electrical resistivity *$[M\Omega\cdot cm]$*	$1.9\cdot10^{9}$	~18	*$\sim10^{10}$*

On the other hand, the main drawbacks could be identified in:

- reduced cooling capability, compared to water;
- compatible materials have to be used for the realization of the cooling circuit; in particular silicone or fluoroelastomer have been successfully used as pump seals; most metals largely used are compatible with perfluorinated fluids; stainless steel, carbon steel and aluminium are commonly used in electronic environment; copper may also be used but may show some discoloration due to surface oxidation;
- care must be taken for the design of the pumping system (suitable hermetic pump with ceramic coating to counteract fluid aggressiveness has been successfully used in other cooling application with perfluorinated fluids [5]).

Some available data regarding fluids radiation hardness [5] have been collected to fully qualify their use for the NBI application.

3. DESIGN OVERVIEW OF THE NBI ION SOURCE

The main requirements of the Ion Source for the ITER NBI are the generation of a beam current a current of 60 A in H$^-$ (and later 40 A D$^-$), accelerated by the voltage of 1 MV. A good beam uniformity and optics are required to match the operating scenarios foreseen for the ITER NBIs.

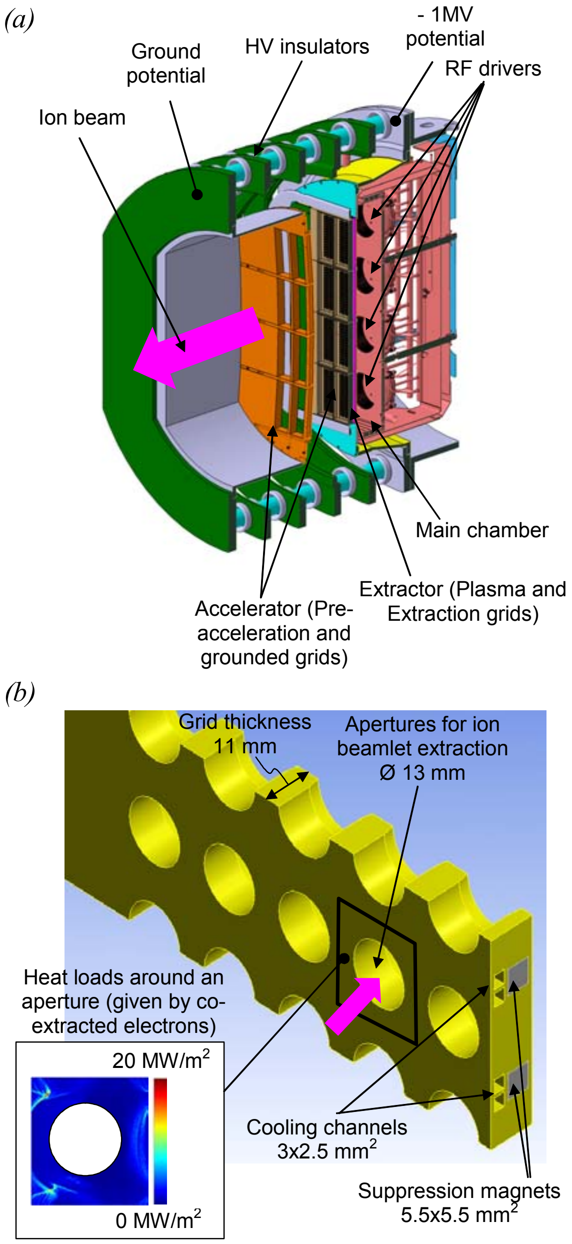

Figure 3. Design of the negative ion source for the ITER NBI (a) Design overview; (b) Detailed section view of the extraction grid

The main components of the Ion Source, sketched in Figure 3a, are:

- A plasma source at -1 MV potential, where the H_2 or D_2 gas is ionized by means of RF current (supplied by eight RF drivers).
- An extractor, composed by the plasma grid at -1 MV (directly in contact with the plasma) and the extraction grid at -990 kV.
- An accelerator, composed by the pre-acceleration grid at -950 kV and the grounded grid at ground potential.

All these components are subjected to heat loads during beam operations. Estimations on these loads come from existing experiments with similar characteristics [6] and from simulations with physics codes [7].

The heat loads represent one of the most critical issues for the design of the Ion Source. In fact the temperature and stress on the hot spots (where the applied power densities are higher) could lead, if the cooling is not sufficient, to undesired phenomena like localized melting of the material, excessive out of plane deformation of the grids, low fatigue life. Hence a sufficiently precise evaluation of the cooling is an important aspect of the design.

The components feature a suitable cooling system with channels running under the most heated surfaces. In order to obtain complicated geometric shapes and to have at same time good mechanical properties, copper electrodeposition technique is used for manufacturing. A rectangular section of the channels is chosen for manufacturing reasons.

The extraction grid (see Figure 3b) represents the most critical component by the thermo-structural point of view, being subjected to very high and concentrated heat loads. Hence, this grid will be considered to compare the cooling efficiency of different fluids.

4. HEAT TRANSFER IN THE GRIDS CHANNELS: DESIGN ISSUES

The evaluation of the heat transfer coefficients in rectangular channels, as described in the literature, is very critical and there is not a clear indication of suitable correlations to be applied with these geometries. The small size of rectangular channels for the plasma and the grounded grid, considered minichannels following Kandlikar and Grande [8] classification, can introduce additional uncertainties in the heat transfer system design. Analytical methods using empirical correlations or computational models solving the Navier Stokes and energy equations can be employed to design one phase minichannel heat transfer systems.

In analytical models, the calculations are strongly influenced by the assumptions on characteristic length and results on heat transfer analysis remain conflicting. Although many studies have been performed, there is not clear evidence that correlations for round tubes can be employed in rectangular channels, both for conventional and reduced size, with satisfactory accuracy. As a consequence, experimental data for geometries similar to the grids ones are scarcely described in literature.

Recently, several authors indicated the employment of numerical solutions to be more suitable in predicting heat transfer coefficients in small rectangular channels than empirical correlations.

Lee et al. [9] found a good agreement between experimental and simulation results suggesting that a conventional computational analysis can adequately predict the heat transfer behaviour in microchannels. They found that numerical simulations with a conventional Navier-Stokes analysis can be employed with confidence for predicting heat transfer behaviour for microchannels of size down to 200 μm. Qu et al. [10] carried out experiments for single-phase water flow in single 222 μm wide and 694 μm deep and 12 cm long rectangular microchannel with Reynolds numbers ranging from 196 to 2215. They found that computational mode shows excellent agreement with pressure drop measurements. They conclude that in the analysis performed the conventional Navier-Stokes equation accurately predicts liquid flow in microchannels and they indicate them as a powerful tool in the microchannel design for electronic cooling.

Arbeiter et al. [11] executed experiments with helium flowing in rectangular minichannels with hydraulic diameter 1.92 mm, for Reynolds numbers up to 12000 to validate the employment of CFD commercial code for the design of high heat flux module. They found that, for the system analysed, the numerical simulations with κ-ε turbulent model are in good agreement with experimental investigations and can be utilized also to predict the local heat transfer.

Viscosity, as reported in [12] can affect significantly the flow field of temperature-dependent fluids and the level of changes depends on channel geometry, fluid type, flow rate, symmetric or asymmetric heating at the wall, heat flux. The influence of varying properties is more critical for dielectric fluids. Shin and Cho [12] numerically investigated the effect of variable viscosity of water and dielectric fluid on the heat transfer and friction factors for laminar flow. They found that the temperature dependent viscosity effect on heat transfer is surprisingly larger for low thermal conductivity fluids. The heat transfer coefficients and friction factor for dielectric fluid are different from those obtained if properties were constant. The analytical approach using constant thermophysical properties could underestimate heat transfer coefficients and overestimate pumping requirements, therefore numerical methods should be preferable to estimate the heat transfer coefficients for dielectric fluids.

5. HEAT TRANSFER ANALYSIS : ANALYTICAL AND NUMERICAL APPROACHES FOR HTC EVALUATIONS

In the present work, cooling calculations have been carried out with CFD tools and analytical methods. These preliminary studies have analyzed one phase flow with Reynolds numbers grater than 10000. A model representing the EG channel and subjected to a uniform heat load was employed to perform the analysis.

Concerning the analytical methods, the approach suggested by Shah and Sekulic [13] for conventional size rectangular channels has been employed: instead of considering hydraulic diameter, a characteristic length has been defined. Blasius correlation has been used to determine friction factor f and Petukhov-Popov correlation to determine the Nusselt numbers. These numbers have been compared with the ones obtained with Gnielinski and Sieder-Tate model for fully developed turbulent flow [14].

For the CFD simulations (performed with the commercial code CFX [15]), in order to verify the influence of turbulence models in the results, calculations with the three most used turbulence models have been carried out.

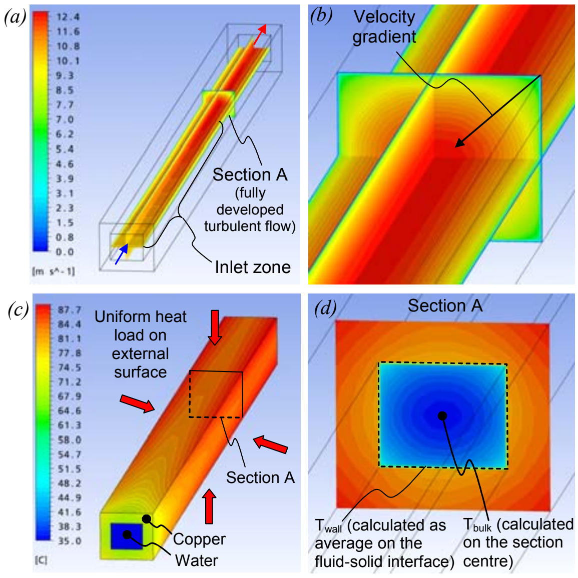

Figure 4. Evaluation of the HTC with numerical methods: (a) Identification of a section (section A) where the turbulent flow is fully developed, i.e. the velocity plot is constant along the flow direction; (b) Detailed view showing the velocity gradient; (c) Temperature plot on the copper and water domains; (d) Temperature contour plot on the section A, with the evaluations of the wall and bulk temperatures.

The k-ε model, where k is the kinetic energy of the turbulence and ε is the dissipation per unit mass, has proven to be stable and numerically robust and has a well established regime of predictive capability. For general purpose simulations, the standard k-ε model offers a good compromise in terms of accuracy and robustness.

The k-ω model, where k is the kinetic energy of the turbulence and ω represents the dissipation per unit turbulence kinetic energy, features a very accurate formulation in the near wall region. The Shear-Stress-Transport (SST) option [16] was chosen in this case, as it is proven to be suitable for the calculation of convective heat transfer [15].

The Reynolds Stress Model (RSM) takes into account the effects of streamline curvature, sudden changes in the strain rate, secondary flows and buoyancy, without using the eddy-viscosity approximation typical of the previous two models [17][18].

For all the simulations, automatic wall functions were used and the mesh was refined in order to give consistent results. In particular the dimensionless wall distance (y^+) was always kept lower than 30, satisfying the recommendations given for this type of models [15]. Moreover a sensitivity analysis, slightly changing the mesh size with the same boundary conditions, was carried out in order to check the results consistency. As the main results are affected by the mesh size only in a negligible way, the chosen mesh is to be considered acceptable and the results consistent.

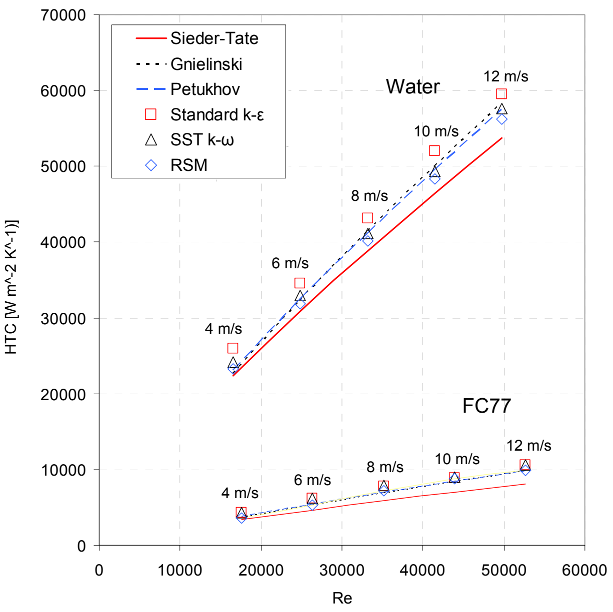

Figure 5. Comparison between CFD and analytical approach for the evaluation of heat transfer coefficient in a rectangular channel (3x2.5 mm^2) with water and FC77 cooling and uniform heat load.

The HT coefficients were in this case calculated with the formula:

$$HTC = \frac{Q}{A_{exchange} \cdot (T_{wall} - T_{bulk})} \tag{2}$$

where the Q is the uniform heat load applied on the channel external side, $A_{exchange}$ is the interface area between water and solid, T_{wall} is the wall average temperature and T_{bulk} is the water temperature in the bulk region.

The water properties were taken from the IAPWS standards [19], where they are evaluated in function of temperature and pressure.
Properties data of FC77 have been calculated with the formula indicated by Incropera [20].

The results are shown in Figure 5. A generally good agreement is found between the different approaches, both for water and the dielectric fluid. The higher discrepancies between analytical and numerical results are found, for both coolants, using the Sieder-Tate correlation.

6. DESIGN OF THE COOLING SYSTEM WITH DIFFERENT FLUIDS

The thermo-mechanical design of the cooling system for the Ion Source grids has been carried out considering the CFD approach. This is expected to give a better accuracy than analytical formulas in the design of the complete grids, where also inlet effects and non–uniform heat loads effects have to be taken into account. Analytical analyses are also considered to asses the results. The HTC are generally underestimated by the analytical formulas and thus, system requirements overestimated.

The cooling data using FC77 has been then compared with those obtained with pure water, using the same boundary conditions and geometry. A simplified model of the extraction grid, representing a slice of the whole grid, is considered to compare water and FC77 in terms of cooling capability.
Heat loads on the extraction grid are due to the co-extracted electrons deviated by the magnets inside the grid (see Figure 3b). The corresponding boundary conditions have been calculated by physical codes in the reference operating scenario of the accelerator and imported as load maps on the grid surface [7]. The resulting peak temperatures on the heated surface and on the channel wall are significantly higher for the dielectric coolant with respect to water calculations, due to the expected better cooling performances of water. The results obtained by the analysis performed with the model previously described seem to not satisfy the alignment requirements. With these heat loads, the employment of FC77 would require changes in the channels geometry, using more space for a larger number of channels in parallel or enhanced channel surfaces.

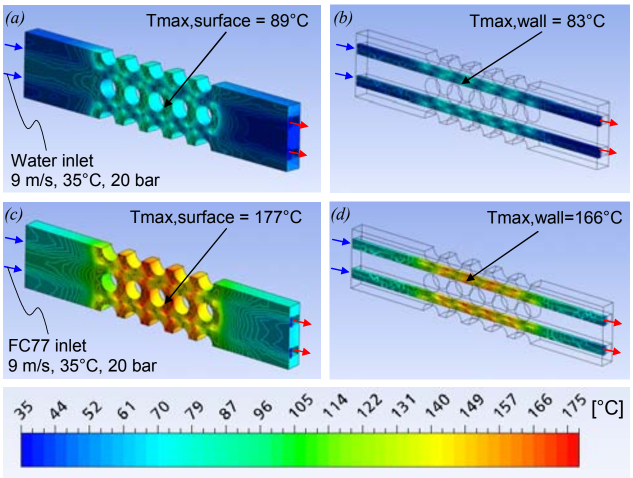

Figure 6. Temperature contour plots calculated with CFD for the Extraction Grid: (a) Surface temperature with water cooling and reference operating scenario; (b) Corresponding wall temperature; (c) Surface temperature with FC77 cooling and reference operating scenario; (d) Corresponding wall temperature

Other High Voltage components, like the plasma grid and the ion source side wall, are subjected to lower values of heat loads and heat power density; in these cases, perflourinated fluids satisfy the requirements with a safety margin and could be used instead of water.

7. CONCLUSIONS

The ITER NBI cooling circuits are required to be able to remove high heat loads and to withstand the electrical insulation between high voltage and grounded components.

Literature surveys and preliminary analyses have been carried out in order to compare water and alternative fluids behaviour. Thank to their remarkable electric properties, the employment of perfluorinated fluids leads to a simplification of the cooling system, with a positive effect on the design.

The cooling capabilities of water and FC77 (one of the perfluorinated fluids, able to work under the required scenarios) have been evaluated using the properties available in literature, with analytical correlations and numerical methods.

The requirements on heat transfer are better satisfied with water, but in this case much care must be taken for the preservation of the water purity in order to maintain leakage current under acceptable level. This implies stringent requirements in terms of water and plant surveillance and water purification system performances.

The analyses demonstrate that the use of perfluorinated cooling fluids would imply major changes to the grids cooling circuit design in order to keep a sufficient cooling efficiency and grids temperature control during the injector operations. Experimental investigations have to be carried out in order to validate the performances of the perfluorinated fluids.

8. REFERENCES

[1] T. Inoue et al, “Design of neutral beam system for ITER-FEAT”, *Fus. Eng. and Design* 56-57 (2001) 517-521.

[2] T. Light, S. Licht, A. Bevilacqua, K. Morash, “The Fundamental Conductivity and Resistivity of Water”, *Electrochemical and Solid State Letters* 8 (1) E16-E19 (2005).

[3] L. Pellegrino, “Experience with long term operation with demineralized water systems at DAΦNE”, *Proceedings of EPAC*, Lucerne, Switzerland (2004).

[4] “3M FluorinertTM Electronic liquid FC-77 Product information” available on website www.3m.com/fluids.

[5] CERN private communication.

[6] E. Speth et al., “Overview of the RF source development programme at IPP Garching”, *Nucl. Fusion* 46 (2006) S220-S238.

[7] P. Agostinetti et al., “Design of a low voltage, high current extraction system for the ITER Ion Source”, *Proceedings of the 1st International Conference on Negative Ions, Beams and Sources*, 2008, Aix en Provence, France.

[8] Kandlikar S.G., Grande W.J., “Evolution of microchannel flow passages - thermohydraulic performance and fabrication technology”, *Heat Transfer Engineering*, 24 (1), pp. 3-17 (2003).

[9] Lee P-S., Garimella S.V., Liu D., “Investigation of heat transfer in rectangular microchannels”, *Int. J. of Heat and Mass Transfer* (2005), 1688- 1704.

[10] Qu W., Mudawar I., Lee S-Y., Wereley S.T., “Experimental and Computational investigation of Flow Development and Pressure Drop in a Rectangular Micro-channel”, *Journal of Electronic packaging*, Vol.128 (2006).

[11] Arbeiter F., S. Gordeev, V. Heinzel, D. Leichtle, E. Stratmanns, “Mini-channel flow experiments and CFD validation analyses with the IFMIF thermo-hydraulic experimental facility (ITHEX)”, *Fus. Eng. and Design*, 2456-2461 (2007).

[12] Shin S., Cho Y., 1996, “Forced convection behavior of a dielectric fluid (FC-77) in a 2:1 rectangular duct”, *Int. Comm. Heat Mass Transfer*, Vol.23, No. 5, pp. 731-744

[13] Shah R. K, Sekulic D.P., 2003, *Fundamentals of Heat Exchanger Design*, John Wiley and sons.

[14] John H. Lienhard, *A Heat Transfer Handbook*, Third Edition, Cambridge Massachusetts, January 2006.

[15] Various Authors, *Ansys CFX 11.0 guide*. Ansys Inc.

[16] F.R. Menter, “Two-equation eddy-viscosity turbulence models for engineering applications”. *AIAA-Journal* 32(8) (1994) 1598-1605.

[17] D. C. Wilcox, *Turbulence Modeling for CFD*. DCW Industries, Inc., 1994.

[18] P. Agostinetti, *Methods for the Thermo-mechanical Analysis and Design of High Power Ion Sources*. Ph.D. Thesis, University of Padua, 2008.

[19] http://www.iapws.org/

[20] Incropera F.P., 1999, *Liquid cooling of eletronic devices by single-phase convection cooling*, Wiley-Interscience Pubblication.