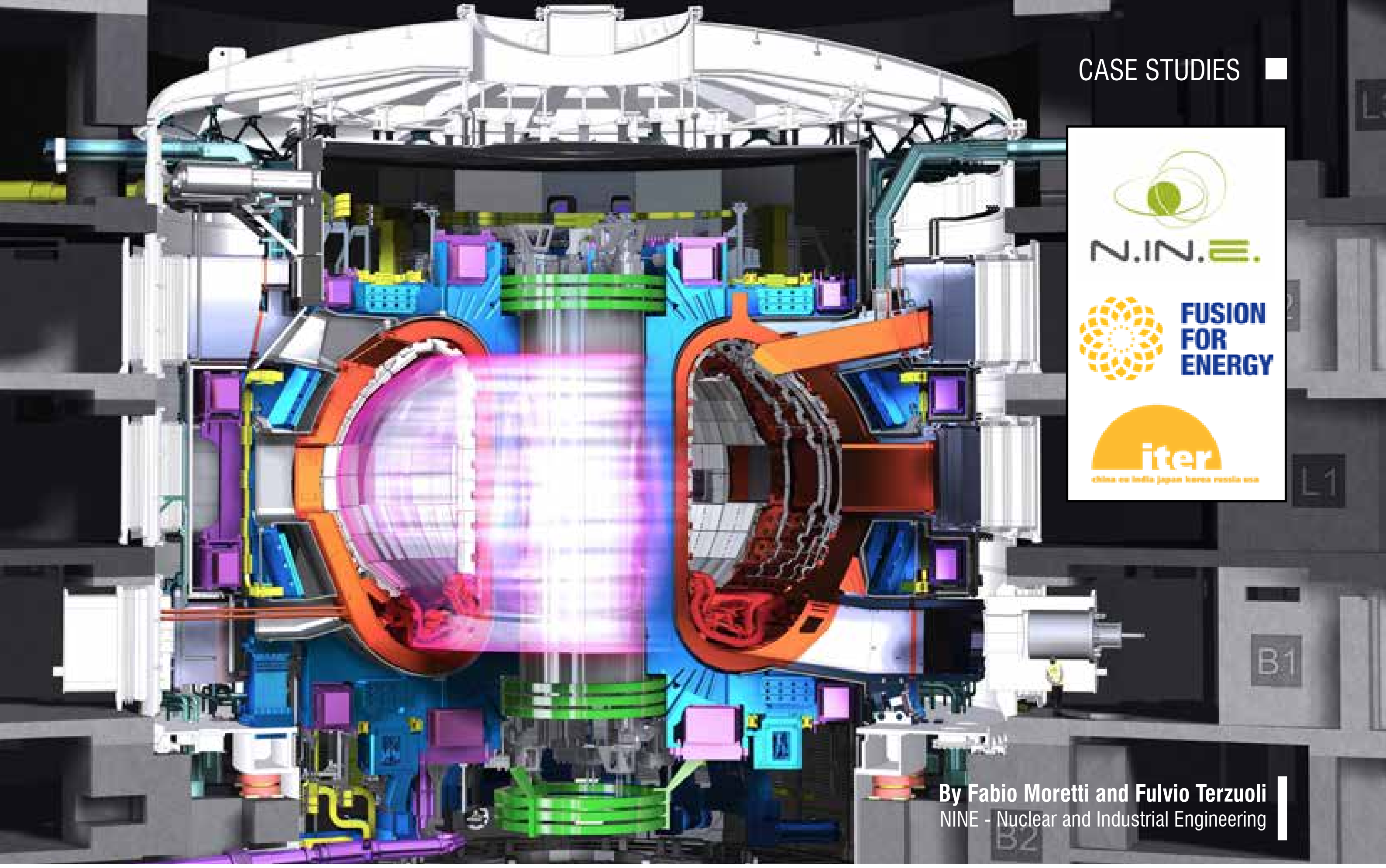

By Fabio Moretti and Fulvio Terzuoli
NINE - Nuclear and Industrial Engineering

Numerical analyses of the heat transfer processes of a key cooling system in the ITER reactor

Supporting the design of a critical system with realistic numerical simulations of heat radiation and conjugate heat transfer

The ITER reactor's electron cyclotron heating and current drive (EC H&CD) system launcher requires an effective cooling system due to the strong thermal loads it supports. In supporting the design of this cooling system, NINE performed several numerical studies using the Ansys simulation tools to:

- *calculate the distribution of the heat flux on the plasma-facing materials by computing the radiation view factors;*
- *conduct a computational fluid dynamics (CFD) conjugate heat transfer analysis of the launcher's structures and cooling circuits, while accounting for thermal loads such as plasma heat radiation, microwave stray radiation, and power deposition from neutron and gamma radiation;*
- *estimate the pressure losses along the cooling circuits.*

These analyses are part of a wider workflow carried out in a collaboration with other partners. The results of the above-mentioned simulations were then transferred to other analysts for the launcher's structural integrity checks.

The International Thermonuclear Experimental Reactor, known as ITER, is an international project aiming to design, construct and operate a tokamak-type (i.e. based on the concept of the magnetic confinement of the plasma inside a toroidal chamber) nuclear fusion experimental reactor, in which the plasma reaches stable physical conditions for a self-sustaining fusion reaction, thus demonstrating that a larger amount of power can be generated than is required to heat up the plasma. The project is being carried out by a consortium composed of the European Union, Russia,

China, Japan, the USA, India, and South Korea. The ITER plant is being constructed near the Cadarache research facility in France. It plans to achieve the so-called first plasma by 2025.
Thousands of scientists, engineers and technicians have been contributing to this huge and ambitious project for decades, facing many scientific, technological, and organizational challenges, generating an enormous amount of progress in many areas, and producing many industrial opportunities. Italian universities, research centers, industries and engineering firms have been deeply involved in the process.

Over the past four years, NINE, a leading Italian consultancy in the design and safety analysis of nuclear installations, has been actively involved, as a third party, in a multi-year framework contract *[the Politecnico di Milano and EnginSoft are also involved in the same project as third parties]* between Fusion for Energy (F4E) *[https://fusionforenergy.europa.eu/]* and NIER Ingegneria Spa to provide support services in the areas of nuclear safety and engineering for the Electron Cyclotron (EC) and the Ion Cyclotron (IC) Heating and Current Drive (H&CD) system antennas (see Fig. 1) of the ITER nuclear fusion reactor project (https://www.iter.org/).

These large and complex devices are intended to help heat and shape the plasma by conveying powerful beams of electromagnetic waves (worth several megawatts) from external generators into the tokamak (see Fig. 2), and are thus exposed to intense thermal loads, such as the heat radiated by the plasma, the neutron and gamma radiation produced by the fusion reactions, and the power deposited by the electromagnetic beams on the transmission lines and various optical components. To give an idea: for the nuclear fusion to occur in the plasma, a temperature of 150 million degrees Kelvin must be reached, and the affected structures of the EC H&CD launcher are only a few meters away from the plasma.

In order to preserve the operational and structural integrity of the reactor and to guarantee safety, all that heat must be effectively removed by dedicated, especially designed cooling systems.

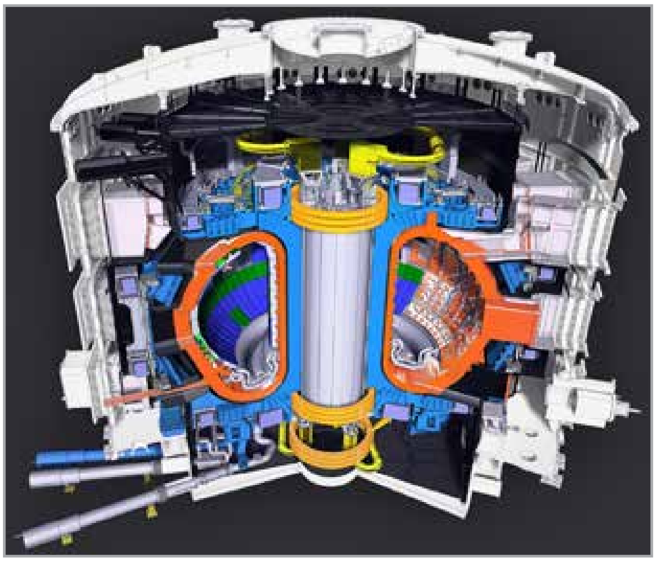

Fig. 2 - The tokamak [Credit ©ITER Organization, http://www.iter.org/]

NINE's focus in particular has been on the heat transfer and thermal fluid dynamic analyses necessary to support the design of these cooling systems. The Ansys simulation tools have been used systematically for this purpose.

Characterization of the thermal loads from plasma heat radiation by calculation of View Factors

One challenging problem that has been successfully addressed is the quantitative characterization of the plasma thermal load on the structures and components of the EC H&CD antenna (or "launcher").

The plasma is contained in a toroidal vacuum chamber and is kept confined by strong magnetic fields, which prevent it from coming into contact with the chamber walls (and with any materials that face the plasma through openings on these walls, such as those belonging to the launchers). Nevertheless, all plasma facing materials are exposed to intense heat radiation, the characterization of which requires solving a rather complex and computationally expensive thermal radiation problem.

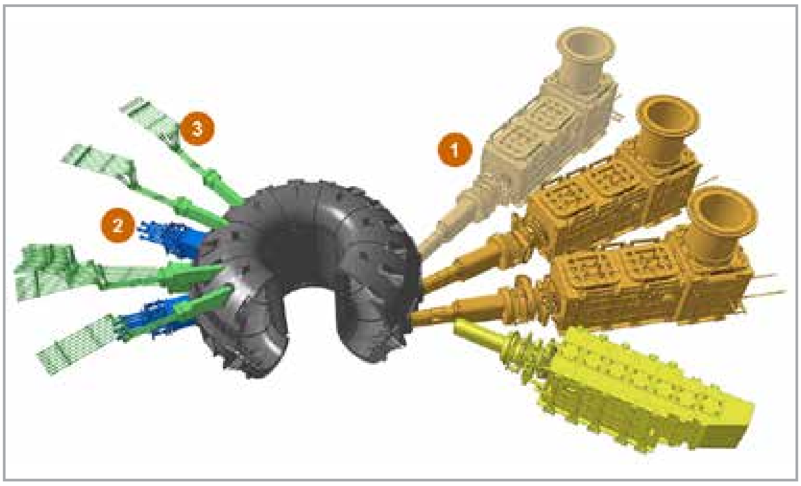

Fig. 1 – Plasma heating external systems: 1) Neutral Beam Injection; 2) Ion Cyclotron Resonance Heating; 3) Electron Cyclotron Resonance Heating [Credit ©ITER Organization, http://www.iter.org/]

Despite the simplifying hypothesis of uniform and isotropic thermal radiation from the plasma, the exposed surfaces are exposed to a heat flux that is distributed in a highly uneven manner, since it depends locally on the surface orientations, their distance from the radiation source, and on the presence of intermediate shielding elements. This type of radiation problem can be reduced to the calculation of the so-called view factors (VF) from the elemental source surfaces to the elemental target surfaces and requires appropriate numerical methods, such as the Radiosity Solution method (and particularly the embedded Hemicube Method) available in Ansys Mechanical APDL, which takes a discretization into shell elements of the entire set of source and target surfaces as input.

The VF calculation, using the Ansys method above, is a computationally intensive task, the cost of which is approximately proportionate to the square of the total number N of shell elements in the model. In fact, a VF factor is calculated for each shell element relative to every other shell element using a robust "brute force" numerical approach.

However, only a small subset of the N(N-1) VFs calculated is usually necessary (i.e., only the VFs from the source shell elements to the target shell elements) and extracting those VFs from the entire set requires additional post-processing steps that can be quite expensive in computational terms if executed by built-in APDL routines.

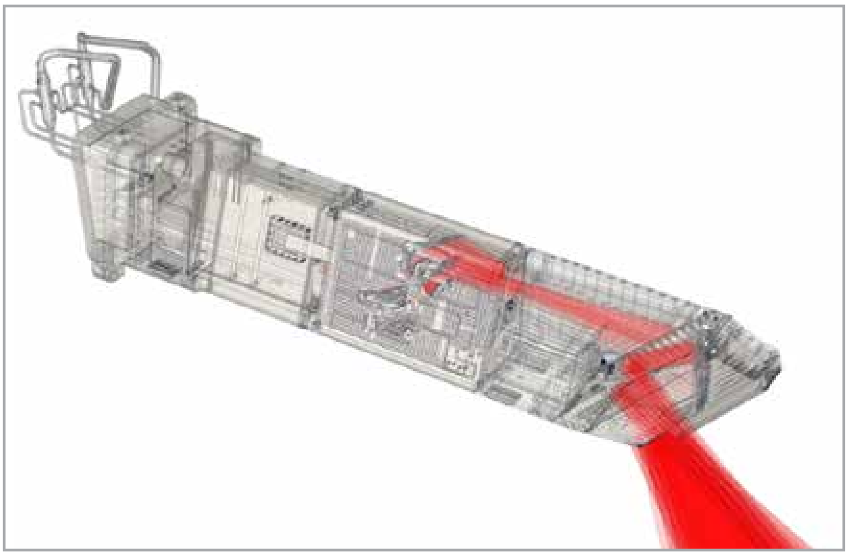

Fig. 3 - One of the four EC H&CD upper launchers [Credit ©ITER Organization, http://www.iter.org/]

NINE has developed its own improved methodology that combines Ansys Radiosity Method for robust VF calculation with ad-hoc Python routines to extract the relevant VFs quickly and efficiently (i.e. with a reduction of processing time by two orders of magnitude).

Once the selected VFs are available, you can obtain the heat flux distribution for the launcher's plasma-facing surfaces and use it as a boundary condition for subsequent heat transfer and structural assessment analyses.

CFD conjugate heat transfer analysis of the port plug cooling system

Another key problem for the structural evaluation was to determine the temperature distribution over the launcher's entire structure, which then had to be transferred to the finite element analysts as input for the thermal-structural assessment.

The geometrical configuration of the launcher's cooling system is extraordinarily complex. The physical quantities that characterize the structure-coolant heat exchange locally (such as heat fluxes and heat transfer coefficients) are spread over very wide ranges, and thus it is not possible to resort to classical engineering correlations without introducing very large inaccuracies into the analysis. Instead, a state-of-the-art detailed analysis approach is necessary, taking advantage of the multi-physics modelling capabilities of CFD codes.

Specifically, a CFD conjugate heat transfer (CHT) analysis is needed, in which the thermal-fluid-dynamic problem (i.e., the solution of the mass, momentum and heat balance equations over the fluid domains that represent the cooling system) is numerically coupled to the problem of the heat conduction through the cooled structures. On the other hand, the use of a coarser two-step approach, involving segregated fluid and solid problems to be solved separately, would require the thermal boundary conditions to be defined (in terms of temperature, or heat flux, or heat transfer coefficient) respectively at the interfaces with solid and fluid domains; in complex cases it can be hard, if not impossible, to obtain reasonably accurate estimates of those boundary conditions, and an analyst may be tempted to make overly simplifying assumptions. In a CFD-CHT analysis those interfaces are handled in an implicit manner (by appropriate continuity and energy conservation conditions) and the distributions of temperature and heat flux are outcomes of the simulation rather than an arbitrary input provided by the user.

International R&D for Carbon-free Energy: the ITER experiment

The International Thermonuclear Experimental Reactor (ITER) is the world's largest fusion experiment. Thirty-five nations are collaborating to build and operate the ITER Tokamak, a magnetic fusion device that has been designed to prove the feasibility of fusion as a large-scale, carbon-free source of energy that uses the same principle that powers our Sun and stars. The ITER reactor has been designed to produce 500MW of output power per 50MW of input.

Just a few meters separate the 4 Kelvin degrees of the cooling liquid in the superconductive coils and the 150-million degrees of the plasma where the fusion reaction takes place. Set between these two extremes, the reactor's metallic surface is exposed to a heating flux comparable to the Sun's: simulation, therefore, plays a key role in the design of the reactor. The ITER Organization and Fusion4Energy, the EU Joint Undertaking for ITER, have committed themselves and through the EnginSoft subcontract, to perform finite element method (FEM) calculations dedicated to different functionalities ranging from the electro-mechanical and magneto-static analysis of the ITER reactor blanket module to the simulation of the welding of the Toroidal Field Coil Cases.

About NINE

NINE – Nuclear and Industrial Engineering – is an engineering company founded in 2011 and based in Lucca, Italy that provides analysis, design, and consulting services mostly in the area of nuclear installations, and to some extent also in the non-nuclear industry, always with a special focus on safety and often within highly-regulated frameworks (such as for nuclear licensing purposes).
NINE's main areas of expertise include: nuclear reactor physics; nuclear reactor system thermal hydraulics; computational fluid dynamics; structural mechanics; uncertainty analysis; fuel behavior; containment system behavior; severe accidents; waste management and radiation protection; probabilistic safety assessment; and so on. NINE's technical staff currently consists mostly of nuclear engineers with a PhD in nuclear and industrial safety. Much of NINE's work involves the use of different types of software to simulate, within a best-estimate multi-physics approach, the behavior of a plant (e.g., a nuclear power station) during various operational scenarios, ranging from normal operation to beyond design-basis accidents. Some of these simulation tools are specifically designed for nuclear applications (e.g., system thermal hydraulic codes, or reactor physics codes, or nuclear fuel performance codes, etc.), while others are general-purpose commercial tools, such as Ansys software for structural and fluid analysis. This expertise can be broadly extended from the domain of nuclear fission to the domain of nuclear fusion, as well as to non-nuclear industrial applications, while keeping the same level of attention to Verification and Validation (V&V), and Quality Assurance.

For further information, please visit NINE's website
(https://www.nineeng.com/) or contact Marco Cherubini
(m.cherubini@nineeng.com).

Case studies can be downloaded from:
https://www.nineeng.com/images/Documents/NINE_CaseStudies.pdf.

The first major step in the CFD-CHT analysis of the launcher was the preparation of the 3D geometrical models of the various necessary solid and fluid computational domains. The source information is constituted by the official ITER 3D CAD models, from which the relevant parts need to be exported (see, for instance, Fig. 3).

As the exported geometry was incredibly detailed and not directly usable for CFD analysis purposes, it was necessary to accurately defeature and simplify all the irrelevant details and to correct several defects. Moreover, the available CAD geometry obviously includes solid parts only, so the fluid volumes had to be "extracted" from them. All these geometry manipulation tasks were efficiently performed by Ansys SpaceClaim, within an Ansys Workbench project.

The outcome consisted of several volumes representing selected "modules" of the launcher's structure and the various sections of the cooling circuits. Those volumes are shown in Fig. 4.

The cyan-colored ones correspond to the cooling circuit for the launcher's structures and consist mostly of channels drilled through the metal bodies and shells that constitute the launcher, which form an intricate network of flow paths.

The volumes in yellow and green correspond to the cooling circuits for the optical components: they must be included in the analysis because, in addition to removing heat from the optical components, they also provide supplementary cooling to certain parts of the launcher's structure.

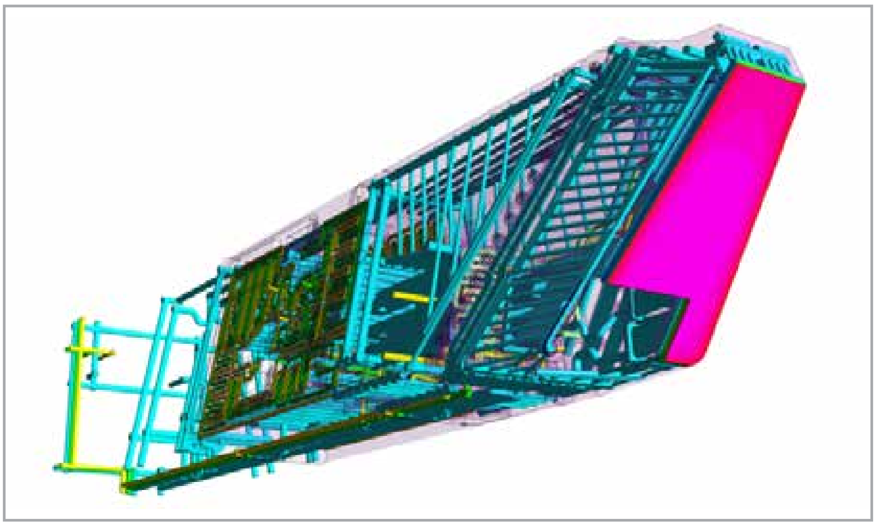

Fig. 4 - The computational domain of the CHT CFD simulations: fluid domains in cyan, yellow and green; solid domain in transparent pink (3 modules of the launcher port plug structure). Heat flux from plasma is contour-plotted on the front end surfaces.

The volumes depicted in transparent pink represent part of the launcher's solid structures, in particular those closest to the plasma (whereas the outermost ones do not play a significant role in the thermal problem and need not be considered in the analysis).

The next key step consisted of the generation of computational grids (or "meshes") for the above volumes with Ansys Meshing. Multiple versions were developed in some cases to allow grid sensitivity studies. The "production" grids, selected on the basis of a balance of accuracy and computational costs, were assembled into a global computational model that counted some tens of millions of nodes and approximately 100 million cells.

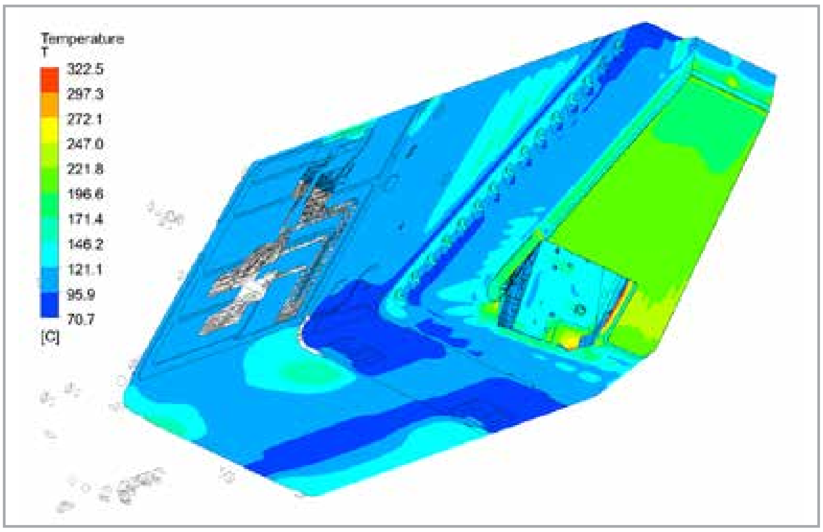

Fig. 5 - Results of the CHT CFD simulation of the EC launcher port plug: temperature distribution over external surfaces

CFD-CHT test and production calculations were then setup in Ansys CFX. Thermal-hydraulic boundary conditions for coolant pressure, flow rates and inlet temperatures were obtained from interface information specified by ITER and F4E and from the results of other in-house analyses.

Thermal boundary conditions for the solid domains consisted of distributions of plasma heat flux (as obtained from the VF calculation task described previously) and of heat flux due to power dissipation from the electromagnetic beams (referred to as stray radiation).

Spatial distributions of volumetric power sources were applied over all domains, to account for the energy deposition from neutron and gamma radiation; those distributions, provided by F4E in the form of "point clouds", were imported into user-defined functions. The Shear Stress Transport (SST) model was used to treat the turbulence. IAPWS-IF97 formulation of water properties, available in CFX libraries, was used.

For the solid materials in the model (stainless steel AISI 316 LN ITER Grade, and CuCrZr alloy) temperature-dependent thermo-physical properties were imported from reference tables provided by F4E. Appropriate thermal contact conductance was applied to those solid-solid interfaces where an imperfect thermal contact takes place (e.g., at bolted connections).

Although the so-called "plasma operation" of the ITER reactor is characterized by a pulsating transient behavior, with the full power conditions being maintained only for a fraction of time in a cycle (e.g., for 600 s of an 1800 s period), the main simulations were performed as steady-state, with stationary full-power thermal loads, thus providing conservative results while maintaining acceptable computational costs.

The steady-state simulations were performed to achieve the best convergence level allowed by the available meshes and the numerical setup used, i.e., with a sufficient number of iterations to minimize the residuals and the imbalances and to stabilize the locally monitored quantities.

In particular, the thermal balances over each domain were carefully checked to verify the correct application of boundary conditions and source terms.

The target results of the main simulation are the 3D distribution of temperature over all fluid and solid domains (Fig. 5), and the distribution of heat flux over all domain interfaces (both fluid-solid and solid-solid). This information was then exported for use as input for the Ansys thermo-structural models by other analysts in the work team.

Further CFD simulations were also performed to estimate the concentrated and distributed pressure losses through the cooling system, thus providing useful quantitative information to support the design and to ensure the fulfilment of the project requirements and interface specifications (such as those for total available coolant flow rate and maximum allowed total pressure loss).

Conclusion

While the scope of these activities is very limited compared to the size, complexity and technological challenges of the whole ITER project, which relies on the collaboration of thousands of engineers and scientists from many countries all over the world, NINE has been able – and continues – to contribute effectively to the advancement of the design of a critical system, i.e. the cooling system for devices that are intended to inject very powerful electromagnetic beams into the plasma to heat it up and help keep it in a stable configuration, in order to reach the conditions for the nuclear fusion reaction to take place.

This is thanks to the skills and experience of NINE's engineers in the area of nuclear safety and design, the powerful simulation tools available in the Ansys suite, the effective collaboration created with the other partners (NIER SpA and Politecnico di Milano), and to the valuable support offered by EnginSoft.

The work leading to this publication has been funded by Fusion for Energy under Framework Contract F4E-OFC-0800.

This publication reflects the views only of the authors, and Fusion for Energy cannot be held responsible for any use which may be made of the information contained therein.

For more information:
Fabio Moretti - NINE - Nuclear and Industrial Engineering
f.moretti@nineeng.com