Fusion Engineering and Design 85 (2010) 1681–1688

Contents lists available at ScienceDirect

Fusion Engineering and Design

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

Progress on an integrated multi-physics simulation predictive capability for plasma chamber nuclear components

A. Ying[a,*], M. Abdou[a], H. Zhang[a], R. Munipalli[b], M. Ulrickson[c], M. Sawan[d], B. Merrill[e]

[a] Mechanical and Aerospace Engineering Dept., UCLA, Los Angeles, CA 90095, USA
[b] HyPerComp Inc., Westlake Village, CA 91361, USA
[c] Sandia National Laboratory, Albuquerque, NM 87185-1129, USA
[d] University of Wisconsin-Madison Engineering Physics Dept., Madison, WI 53706, USA
[e] Idaho National Laboratory, Idaho Falls, ID 83415, USA

ARTICLE INFO

Article history:
Available online 31 May 2010

Keywords:
Blanket technology
ITER
Design analysis
Simulation

ABSTRACT

Understanding the behavior of a plasma chamber component in the fusion environment requires a simulation technique that is capable of integrating multi-disciplinary computational codes while appropriately treating geometric heterogeneity and complexity. Such a tool should be able to interpret phenomena from mutually dependent scientific disciplines and predict performance with sufficient accuracy and consistency. Integrated multi-physics simulation predictive capability (ISPC) relies upon advanced numerical simulation techniques and is being applied to ITER first wall/shield and Test Blanket Module (TBM) designs. In this paper, progress in ISPC development is described through the presentation of a number of integrated simulations. The simulations cover key physical phenomena encountered in a fusion plasma chamber system, including tritium permeation, fluid dynamics, and structure mechanics. Interface engines were developed in order to pass field data, such as surface deformation or nuclear heating rate, from the structural analysis to the thermo-fluid MHD analysis code for magnetohydrodynamic (MHD) velocity profile assessments, or from the neutronics analysis to the thermo-fluid analysis for temperature calculations, respectively. Near-term effort toward further ISPC development is discussed.

1. Introduction

The behavior of a plasma chamber nuclear component in the fusion environment is complex. Understanding and predicting its performance requires many specialized computational codes, such as Monte Carlo N-Particle (MCNP) code for neutronics, computational fluid dynamics (CFD)/thermo-fluid codes for first wall (FW) surface and structural temperatures, structural codes for stress/deformation analysis, and mass transport modeling for tritium permeation estimation. Because of the interdependent nature of these phenomena, it is essential to apply an integrated multi-physics simulation capability. The need for an integrated simulation predictive capability (ISPC) is further increased by the complex geometry of the fusion system. It has been demonstrated that these analyses should be performed in 3D, with a true geometric CAD representation, in order to achieve high-quality predictions [1]. Including geometrical details into the analysis is crucial in predicting performance with a reasonable level of accuracy. An ISPC that can handle the complicated geometry of manufactured components can substantially reduce the risk and cost of developing the complex multi-dimensional system comprising the plasma chamber in-vessel components. During the conceptual design phase, design analysis can help to optimize and increase design margins. During the engineering design phase, design analysis can eliminate a significant number of design flaws. The process of ISPC development will also serve to integrate results of ongoing R&D and will ultimately provide a validated predictive capability for DEMO.

A multi-physics simulation model, centering on CAD, has been proposed and launched in the ITER first wall (FW)/shield blanket and ITER Test Blanket Module (TBM) design activities [2–6]. This paper presents and discusses progress toward the development of ISPC, followed by a short description of the framework of ISPC. Recognizing that the scientific and technical challenges in constructing automated, CAD-integrated, high-computing-performance, multi-physics simulation suites are significant, this paper addresses what we can achieve and how it can be achieved. It also addresses where the interfaces may be, and demonstrates the importance of this capability in designing engineering components for the machine, such as ITER and ITER TBM. A grant effort to present this predictive capability in a virtual world requires a much broader expertise,

* Corresponding author at: Mechanical and Aerospace Engineering Dept., UCLA, Room 44-136, Engineering IV, Los Angeles, CA 90095, USA. Tel.: +1 310 206 8815; fax: +1 310 825 2599.
E-mail address: ying@fusion.ucla.edu (A. Ying).

0920-3796/$ – see front matter © 2010 Elsevier B.V. All rights reserved.
doi:10.1016/j.fusengdes.2010.05.015

Nomenclature

$\bar{\rho}$	mean value of density
$\bar{C}$	mean value of species concentration
$\dot{C}_s$	species generation rare
x_j	coordinate
$\bar{u}_j$	mean velocity in j coordinate

including computer and mathematical scientists, and has yet to be launched.

1.1. The framework of ISPC

The development of ISPC relies upon advanced numerical simulation techniques that are capable of handling geometric heterogeneity and complexity. The ISPC has been enabled by the ability to perform all analyses on geometric models that are derived from an identical representation, i.e., the CAD-based solid model, as illustrated in Fig. 1. The codes that simulate key physical phenomena of the plasma chamber systems form the first component of the ISPC. These codes solve governing equations characterizing the disciplines of neutronics, electro-magnetics, plasma–material interaction, thermo-fluids, species transport, and structural mechanics. Likewise, the candidate codes have to show their capabilities to interface with complex, three-dimensional, prototypical-sized CAD models. Effort is ongoing to assemble a combination of fusion-specific research codes and existing, mature, off-the-shelf software. The computational codes currently under evaluation and development for use in the ISPC application are listed in Table 1. These codes must be validated, enhanced, and tuned in order to be applicable to the fusion plasma chamber environment. For example, constitutive equations describing the thermal–physical properties of a heterogeneous medium of a breeder pebble bed need to be added to the existing codes. These properties, as well as critical local phenomena, can be modeled in a finer domain, while calculated results can be mapped and transmitted to the global realm for incorporation. Off-the-shelf CFD codes cannot analyze the magnetohydrodynamics (MHD) phenomena occurring in a liquid breeder cooled blanket such as a DCLL (dual-coolant lead–lithium). Consequently, a fusion-specific liquid metal MHD code has to be developed and incorporated into the integrated simulation suite. The advancement of capabilities to model fusion-specific phenomena, such as MHD and plasma facing component (PFC) melt layer dynamics, is also foreseen as an important step of ISPC development.

The second component includes data translation, involving efficient and high-fidelity mapping or interpolation of data across various analysis codes to enable integrated or coupled simulations in multi-physics and multi-scale environments. This is because, in most realistic calculations, the computational meshes used for different physical analyses are very different in nature. Numerical modeling of individual physics has its own unique mesh resolution requirements, and thus data translation fill in. Data translation interfaces have been developed to map the MCNP calculated nuclear heating rates at structural grid tallies to the centroid nodes of the CFD tetrahedral mesh elements for thermo-fluid calculations [2]. Furthermore, the modeling of accident scenarios, in particular, may require a detail simulation of the effects of the entire system on an individual component. This calls for a simulation framework with a hierarchy built into it, so that codes that perform global, system-level modeling can communicate with codes that carry out detailed, component-level modeling. Development of this hierarchical framework will be a part of ISPC.

The third component of ISPC is computational analysis management, including execution control, deformed geometry treatment, and new mesh generation. MHD flows at high Hartmann number are extremely sensitive to changes in design. It is conceivable that both static features (such as total pressure drop), as well as dynamic or unsteady flow phenomena, could be influenced by factors such as structural deformation due to pressure/thermal loads (Fig. 2). We have explored non-uniform rational basis spline (NURBS) based interpolation techniques relating mesh-based data to the CAD model. The geometric deflection at each mesh point and its deflections (dx, dy, and dz) are calculated from a struc-

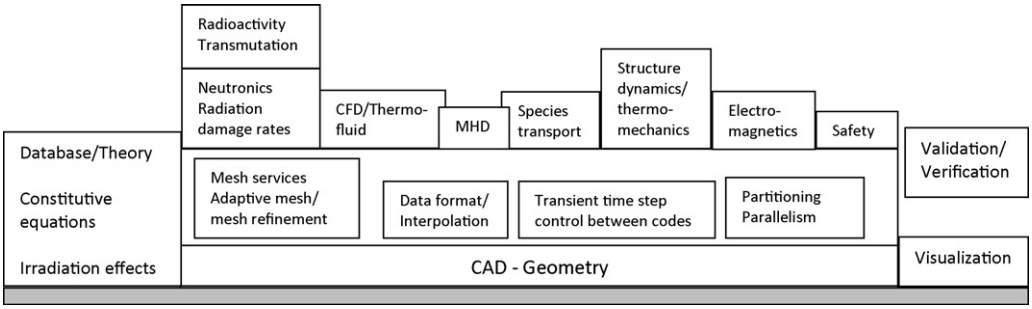

Fig. 1. The framework and components of the ISPC.

Table 1
ISPC uses a combination of fusion-specific research codes and off-the-shelf third-party software.

Physics	Analysis code	Mesh specification
Neutronics	MCNP	Particle in cell (PIC)
	Attila	Unstructured tetrahedral mesh (node based)
Electro-magnetics	OPERA (Cubit)	Unstructured tetrahedral (Hex-) mesh (node based)
	ANSYS	Unstructured Hex/Tet mesh (node based and edge based formulations)
CFD/Thermo-fluids	SC/Tetra	Unstructured hybrid mesh (node based)
	Fluent (Gambit)	Unstructured hybrid mesh (cell based)
MHD	HIMAG	Unstructured hybrid mesh (cell based)
Structural analysis	ANSYS/ABAQUS	Unstructured second order Hex/Tet mesh (node based)
Species transport	COMSOL	Unstructured second order mesh (node based)
Safety	RELAP5-3D MELOCR	System representation code

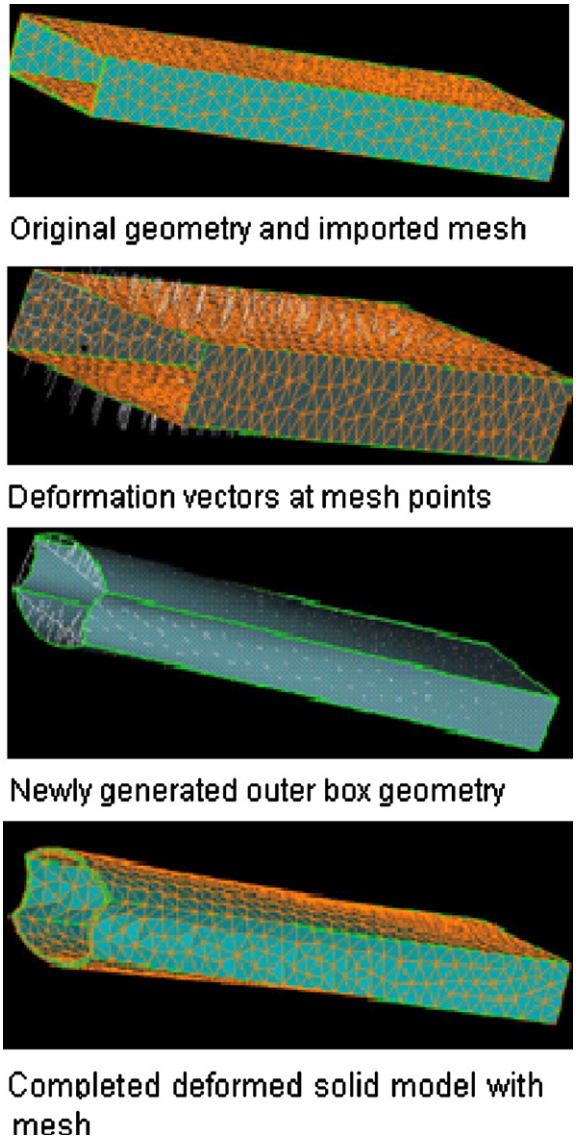

Fig. 2. A CAD-grid mapping technique being developed to transfer a deformed structure to HIMAG MHD meshes to study the effects of structure deformation on MHD flow.

tural analysis. For each point d that defines the NURBS curve, the mesh element and associated vertices containing d can be located through an octree search algorithm. The deflection can be mapped into the NURBS coefficients, and the surface can be deformed, permitting a new mesh to be generated on the deformed surface. The deformed surface shapes can then be used to regenerate a volume mesh for each material region [7]. The same algorithm can be used to interpolate data between solver modules, such as fluid mechanics, heat transfer and structural analysis modules, even when no deformation is present. The ISPC also serves the role of a simulation-process-management system, wherein all of the data relevant to each aspect of the simulation is stored and transmitted to multiple solvers in appropriate formats, as well as made available for post-processing and debugging. Post-processing, extraction of appropriate data, preparing time evolutions of key performance parameters, and being able to export graphical information for visualization are all important functions of ISPC.

Validation and verification form the fourth component of the ISPC. Validation is needed at all levels, including validation of individual codes, execution procedures, data structures, and data translation techniques. Experimental data will be needed to validate both individual physical models and the integrated model. ITER and other integrated fusion testing facilities should generate benchmark data for ISPC validation, especially on fully integrated component-level performance in the fusion environment.

2. Application to design verification

Successful design and development of a complex system like the ITER TBM demands a complete three-dimensional CFD/thermo-fluid analysis with an integrated calculation of the flow field (in the fluid components of the TBM) and the temperature field (in both the fluid and solid components). One candidate CFD code is SC/Tetra, a computational thermo-fluid analysis system based on an unstructured hybrid mesh finite-volume method. The software suite includes an automatic mesh generator capable of handling the complex geometry of an as-manufactured component. Such a tool is essential to reduce design uncertainties, particularly in designs where the coolant flow distribution is relatively complicated and involves many parallel flow paths. However, a significant challenge arose in creating a workable CFD model, which requires the underlying CAD model to have zero intersecting faces. These intersecting faces occur in the CAD model as tangential contacts between two surfaces, existences of imperfect surface duplications, and/or tiny gaps. Removing the intersecting faces in the CAD model was a very time-consuming task.

As a test case, the complete CAD model of a dual-coolant lead-lithium (DCLL) TBM design was loaded into the SC/Tetra CFD code. The lead–lithium (PbLi) flow circuit was modeled as an ordinary fluid, without considering MHD effects. Neglecting MHD effects does not affect the helium fluid flow or structural temperature calculations, since lead–lithium is thermally isolated from these elements. The calculation provides the steady-state temperatures in the solid and fluid parts, as well as the complete 3D flow distribution of He coolant in the manifolds and in the coolant channels.

The initial model shows that helium enters the TBM through an inlet pipe located near the bottom plane. It is then divided into 2 parallel flow paths, moving upward sequentially in groups, with 8 channels per group to cool the entire first wall. Each parallel path supplies four channels per group through a supply manifold; however, as shown in Fig. 3 (the top two pictures), the flow is gradually reduced as it moves away from the injection location. This implies that the supply manifold is not large enough to equalize the pressure drop. The helium flow is relatively non-uniform among the channels within a group, and becomes tremendously reduced near the top of the TBM (Fig. 3, bottom left), which leads to unacceptably high surface temperatures. This reduction in flow occurs because a portion of the flow is directed to cool both the divider and the upper plates, due to the fact that their entrance locations are in the line of the flow path. These modeling results are being used to modify the coolant distribution scheme in order to achieve a uniform coolant flow, as well as to maintain the maximum temperature within operational limits.

MHD modeling shows that the liquid metal (LM) MHD flow characteristics are strongly driven by the temperature gradient of the LM. Depending on the location of the LM, the flow velocity induced by natural convection can be greater than the MHD velocity by forced convection. The consequences of natural convection include increases in the interface temperature and heat leakage, and reverse flow. An MHD flow analysis, integrated with heat transfer and buoyancy analyses, was performed for a DCLL-type duct with an enclosed flow channel insert made of low conductivity material using HIMAG code. As seen in Fig. 4, there appear strong recirculation currents caused by natural convection, and there are flow stagnations near the top and bottom corners of the channel. Without integrating heat transfer with the MHD fluid flow

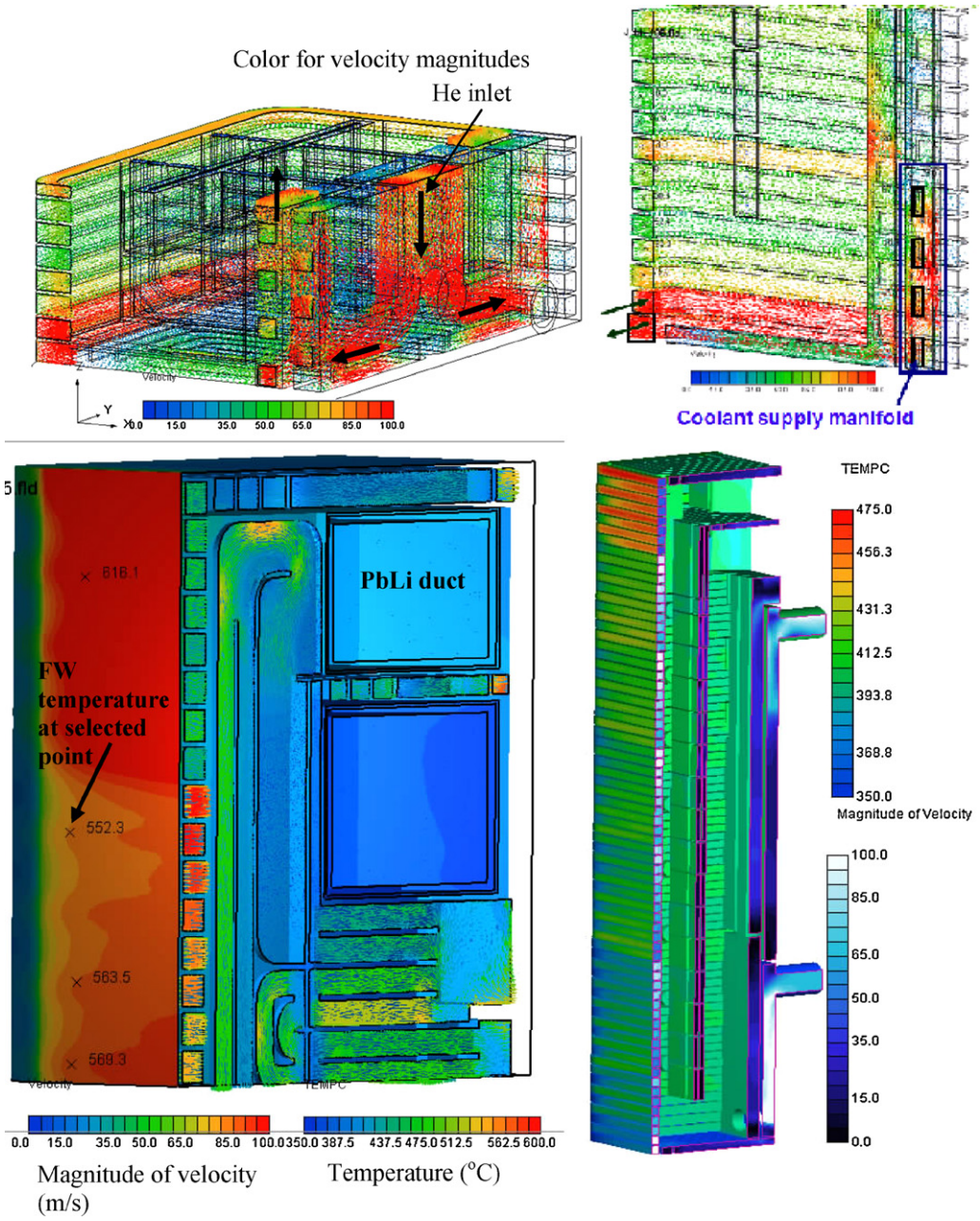

Fig. 3. Snapshots of temperature and flow characteristics of the DCLL He-circuit. Top left: He inlet stream is divided into 2 parallel paths among the coolant channels. Top right: He flow characteristics in a He inlet flow distributor. Bottom left: a large non-uniformity in the flow found in the top of the He channels. Bottom right: wall/He interface temperature and flow distribution of FW He channels.

calculations, ensuing research and development (R&D) activities might be inadequately prescribed.

The temperature and hydraulic pressure field results in the solid components were then imported as loading conditions into the structural analysis system (for example ANSYS structural code). Typically, a CFD system is based on the finite-volume method, while a stress analysis system is based on the finite element method. One method of coupling CFD and stress analysis codes is to convert temperature field data from the CFD code to body load data in the stress analysis code. Similarly, static pressure load in CFD code can be converted to surface load in the stress analysis code. Fig. 5 shows the CFD temperature field data for the DCLL ferritic steel structural box loaded into ANSYS, and the corresponding calculated stress magnitudes. This method is advantageous if the geometry under analysis is very complicated, and if a fine resolution is required in different parts of the geometry in the CFD stage versus the stress analysis stage. For example, when setting up the thermal analysis in the structural analysis code, particularly for a transient case, it is necessary not only to define the film heat transfer coefficient, but also the ambient temperature for each individual surface node, in order to arrive at an actual bulk temperature at solid [8]. One advantage of the computational mesh constructed by SC/Tetra is its ability to mesh solid and fluid domains separately. For example, the model can be generated with only tetrahedral elements in the solid thermal-stress domain and with a hybrid mesh (a mixture of tetrahedrons, prisms, hexahedrons, etc.) in the fluid domain. Because the hybrid mesh and variables on nodes are compatible between structural analysis FEM elements and node-based CFD elements, the temperature distribution of not only the solid, but also of the mesh, can be directly applied to the thermal-stress analysis. On the

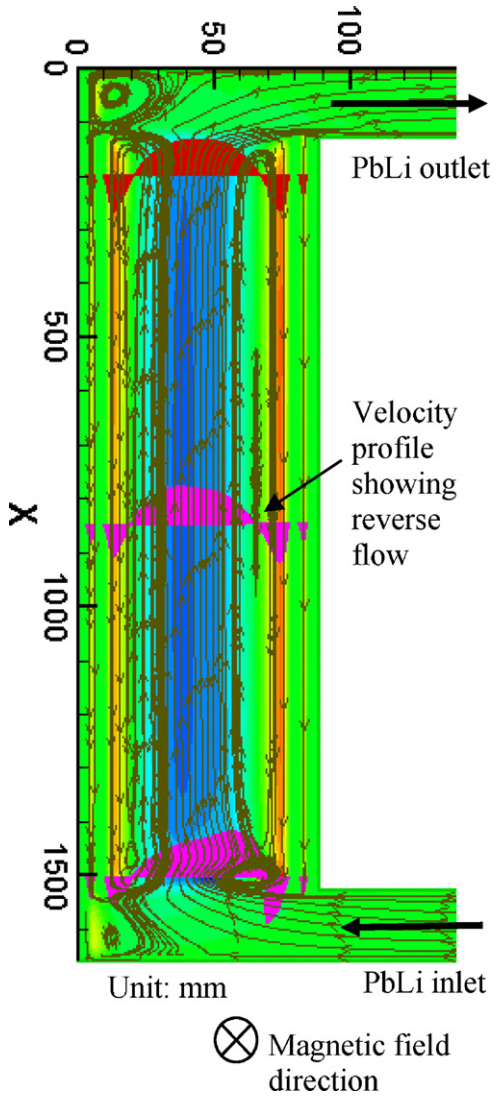

Fig. 4. Streamlines revealing complex PbLi MHD flow characteristics in a typical DCLL channel under nuclear heating.

other hand, the temperature field data can be directly mapped to structural meshes that were generated by the finite element structural analysis code. The ability to use separate, tailor-made meshes for the thermo-fluid and thermal-stress calculations, by the use of data mapping between the two computational codes, provides an effective means for developing ISPC.

3. Predicting nuclear performance to a reasonable level of accuracy

One objective of the ITER TBM testing is to provide data about tritium generation in a DEMO look-alike blanket design, in order to evaluate D-T fuel self-sufficiency. In addition, data about the exact distribution of nuclear heating in the components will be an important input for subsequent CFD and MHD simulations. To pin point where monitoring instrument should be located, and to know quantitatively the disturbance in the data that will be caused by the instrumentation, a detailed neutronics analysis, including all geometric and composition details, becomes a prerequisite for any meaningful ITER TBM nuclear experiments. This is evident from a recent 3D neutronics analysis of a DCLL TBM, where the surrounding water-cooled frame and other in-vessel components are included in the calculation [1]. The DCLL TBM is geometrically complex and is composed of multiple materials. To account for these complexities, the DAG-MCNP code [1,9] was used to perform 3D nuclear analysis of the TBM. Because neutrons ejected from the plasma will slow down as they pass through the water in the water-cooled frame, and because slower neutrons more efficiently breed tritium in Li-6, the calculated tritium production rate is higher at the edges of the module (Fig. 6). The total tritium generation rate in the PbLi is 4.19×10^{-7} g/s during a D-T pulse with 500 MW fusion power, which is approximately 45% lower than the value obtained from 1D estimates [1]. The calculated total nuclear heating in the TBM is 0.374 MW. It is interesting to note that this is 35% lower than that estimated based on 1D models [1]. These findings demonstrate the need to account for all geometrical details and heterogeneities in order to accurately predict of nuclear parameters.

Much progress has been made recently in adapting MCNP to complex geometry situations, which allows for the reduction of human error, particularly as geometrical complexity increases. In DAG-MCNP, the CAD model is read and evaluated directly by the Monte Carlo code. This approach provides a number of advantages, including the ability to represent surfaces beyond the limitations of the native MCNP geometry, and the introduction of a common domain representation that facilitates coupling to other physics simulations.

The fundamental geometric operation in a Monte Carlo radiation transport code is to test the distance between a point and a surface, along a particular ray. The DAG-MCNP approach uses the mesh-oriented database (MOAB) [10] and Common Geometry

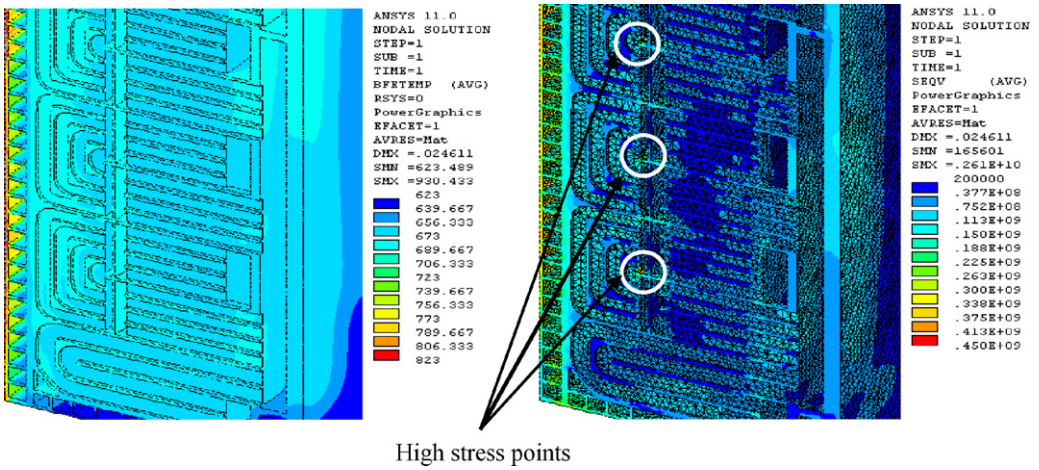

Fig. 5. Temperature field data from the CFD stage, loaded into structural analysis meshes for stress assessment. Several high stress concentration locations were identified, which need to be ameliorated in the next design.

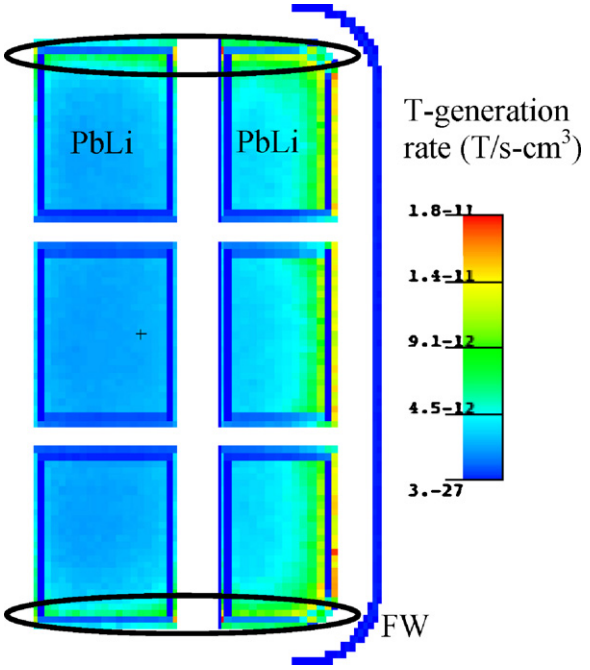

Fig. 6. Tritium production profile at the DCLL mid-plane. The high values near the edges (circled regions) were caused by the stainless steel water frame.

Module (CGM) [11] software libraries to provide this ray-surface intersection capability at the surfaces of the CAD-based solid model. The overwhelming majority of execution time in a direct geometry approach is taken up by the ray tracing used to track particles as they travel through and interact with the domain. Several methods have been developed to accelerate this calculation. The most significant of these accelerations is to perform facet-based ray tracing. The resulting code runs at speeds approximately 2–4 times faster than standard MCNP, with negligible loss of accuracy in typical cases. Ref. [1] gives detailed description of these accelerations.

The workflow includes several steps. The initial step is to construct the geometrical model in a CAD-based system. It is essential to ensure that the model is "clean", with each point in the space uniquely defined in a single volume. This might require some repair for overlapping volumes, or to establish proper contact between adjacent volumes. Another part of the geometry preparation is to eliminate duplication of surfaces between adjacent volumes in the model [9]. The result is a series of contiguous volumes, separated from each other by surfaces each shared by two volumes, with remaining surfaces bounding a single volume, with void space on the outside. This makes finding the volume entered after crossing a surface a topological check, which is much more efficient than ray tracing. The most complicated and tedious step in preparing geometry for analysis is the generation of the complement, or void space, which surrounds all the solid/liquid objects in a model. An approach in which the complement is represented implicitly has been developed and is a standard feature of DAG-MCNP [9].

4. Integrated simulation for tritium permeation analysis

In most realistic calculations, numerical modeling of separate physics involves separate numerical schemes and computational mesh resolutions. However, if the numerical techniques for the different physical analyses (e.g., heat conduction and species diffusion) are analogous, it is advantageous to extend the code's capability to solve both physical phenomena and therefore avoid transfer and interpolation of field data across codes, and the resultant risk of introducing error. It is feasible to extend thermo-fluid code to also perform tritium transport analysis. This is particularly crucial for accurate prediction of the rate of tritium permeation from the breeding zones to the coolant. Tritium permeation characteristics and quantity are governed by multi-physics phenomena, including a convection term from the fluid flow, temperature-dependent diffusion coefficients, tritium production profiles, and multiple tritium species. The SC/Tetra CFD thermo-fluid code was chosen for evaluation because of its superior speed and large-scale simulation capability. SC/Tetra solves the diffusive species conservation equation. Additionally, it can model chemical reactions and flow in porous media. These capabilities serve as the basis for analyzing the tritium permeation rate, and for designing the purge gas system in a solid breeder blanket.

A finite-volume formulation is used in the SC/Tetra solver to discretize the governing equations based on the cell vertex formulation. During the execution, zero-thickness gap elements are inserted at the material interfaces. Inserting gap element layers places one extra node at the exact location of each node along the boundaries between two different materials, making the mesh discontinuous across the boundaries. The gap elements define the movement of heat and of species between the elements on both sides. The equation to be solved for conservation of diffusive species in a turbulent flow is expressed in terms of the mean values and the perturbations thus:

$$\frac{\partial \bar{\rho}\bar{C}}{\partial t} + \frac{\partial \bar{u}_j \bar{\rho}\bar{C}}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\bar{\rho} D_m \frac{\partial \bar{C}}{\partial x_j} - \bar{\rho} u_j' \bar{C}'\right] + \bar{\rho}\dot{C}_s \quad (1)$$

where D_m is as the mass diffusivity coefficient, and is usually a tensor; here, it is assumed to be isotropic.

$\bar{\rho} u_j' \bar{C}'$ is the amount of diffusive species transported by turbulence and is modeled as

$$\bar{\rho} u_j' \bar{C}' = -\bar{\rho} D_{mt} \frac{\partial \bar{C}}{\partial x_j} \quad (2)$$

where D_{mt} is the turbulent diffusion coefficient, and can be obtained using the turbulent Schmidt number (0.9) and the eddy viscosity obtained from respective turbulence model.

In this initial development, only equilibrium tritium transport physics is considered. Further effort is needed to account for tritium kinetics. The boundary conditions to be applied at the solid/liquid interface under the equilibrium conditions include: (1) using Sievert's law for equilibrium concentration estimation at the solid face of the interface, in which the discontinuity of the concentration profile is simulated; (2) ensuring a continued diffusive flux at the interface. During the flow solving process, prism layers were inserted into the fluid wall boundaries to simulate viscous sublayer flow phenomena. In executing the diffusive flux continuity boundary condition, a prism layer was also inserted on the solid side of the solid/fluid interface. This approach requires no additional treatment for calculating and extracting the surface area of the corresponding solid and fluid faces at the interface. This capability has been applied to designing the helium purge gas system in a solid breeder unit which involves a turbulent helium coolant flow in rectangular channels, porous purge gas flow characteristics in the breeding zone, temperature and tritium species diffusion and transport calculations over the breeding zone, coolant containing structures, and coolant flow. An example tritium concentration profile across various regions of the breeding zone is shown in Fig. 7. Here, a purge gas is injected near the top of the breeding zone and close to the FW, and is removed at the bottom, near the back. Fig. 7 shows that tritium appears to accumulated at the bottom front of the breeding zone, and to be more diffuse at the hot side of the structure. This may require that the injection point of the purge gas should be moved toward the mid-plane of the breeding zone. Additional details of this work including a benchmark anal-

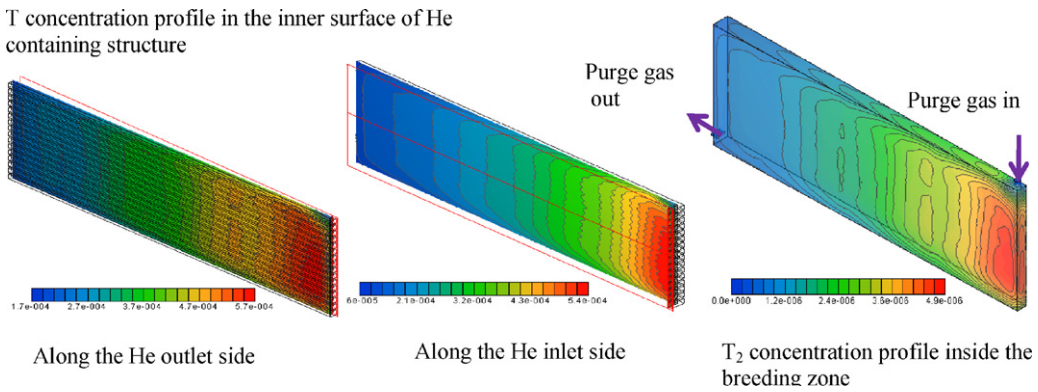

Fig. 7. Calculated tritium concentration profiles in various parts of a solid breeder unit (The non-symmetrical purge gas flow results in a higher tritium concentration at bottom part of the breeding zone. Tritium concentration profiles in the containing structures reflect this effect.).

ysis with the experimental data from reference [12] are presented in a companion paper [13].

5. Multi-scale simulations add critical insights

While finite-difference or finite element numerical-scheme-based computational codes can provide a basis for describing most of the chamber component physics, it is still necessary to integrate mini-scale discrete element and/or system analysis codes for enhanced understanding of the phenomena involved, such as the ceramic breeder blanket pebble bed thermo-mechanics and primary coolant flow transient behavior.

5.1. Integration between discrete element and continuous model for pebble bed thermo-mechanics

Maintaining the integrity and thermo-mechanical state of the ceramic breeder pebble bed throughout its lifetime is crucial to the success of ceramic breeder blanket designs. In a typical design analysis, the stress and strain generated by the differential thermal expansion between the pebble bed and its containing structure can be estimated using a continuum model. Such a model uses structural analysis code like ANSYS, incorporating experimentally derived thermal–physical and mechanical properties. Strictly speaking, the pebble bed cannot be viewed as a solid; the pebbles can behave like fluid, deforming or sliding away from their original positions under compressive loads. To handle particle movement and assess the consequence of the generated stress on the pebbles' integrity, a discrete element model (DEM) can be employed [14,15]. Efforts are under way to integrate macroscopic stress and strain magnitudes that are calculated by a continuum model, as the mechanical boundary conditions for the estimation of the microscopic inter-particle contact force by means of DEM simulation.

5.2. A 1D–3D CFD/thermo-fluid integrated simulation

The internal flow behavior of a plasma chamber component simulated by a 3D CFD code is affected by its flowing system network components (i.e., pump, heat exchanger, etc.) typically modeled by 1D (or simplified 3D) system code. A 1D–3D co-simulation can more realistically model internal thermo-fluid behavior while eliminating uncertainties in the boundary conditions assumed in the 3D model. Example problems concern FW temperature evolutions under a loss of flow transient and/or heat exchanger operational scenarios under ITER pulse operations. Regarding the second problem, a question arises: how would the heat exchanger be designed to follow an ITER inductive pulse in which, within a 1800 s burn, the plasma stays on for only about 400 s and off for more than 1100 s? Under such an operation, should the coolant inlet temperature be maintained at a constant value over the course of each pulse, with the heat exchanger providing residual heat removal during plasma off? Or should the FW/blanket temperature profile be kept at a quasi-steady state to avoid thermal fatigue by by-passing the water from the heat exchanger? The water outlet temperature evolutions during an inductive pulse are plotted in Fig. 8 for each of these two scenarios. In the first scenario, with the heat exchanger being in a stand-by mode, the outlet temperature increases during plasma off, but returns to its steady state condition if the inlet temperature can be brought back to its normal operational during plasma on. In the second scenario, the water outlet temperature decreases during plasma off if the inlet temperature is maintained at its original state. These preliminary calculations have not considered the time delay caused by the water circuit, which could be estimated by RELAP5 or a similar code. To capture the effect of the flow circuit on an engine cooling problem, a co-simulation methodology has been developed using CFDLink for integrating 1D Flowmaster CFD code with 3D SC/Tetra fluid flow models [16]. In this technique, the data exchange between 1D and 3D is automatically performed in an arbitrary iteration step, which allows much more data exchange and higher accuracy. Efforts are underway to implement similar techniques for integrating MELCORE and/or RELAP5 with a 3D CFD

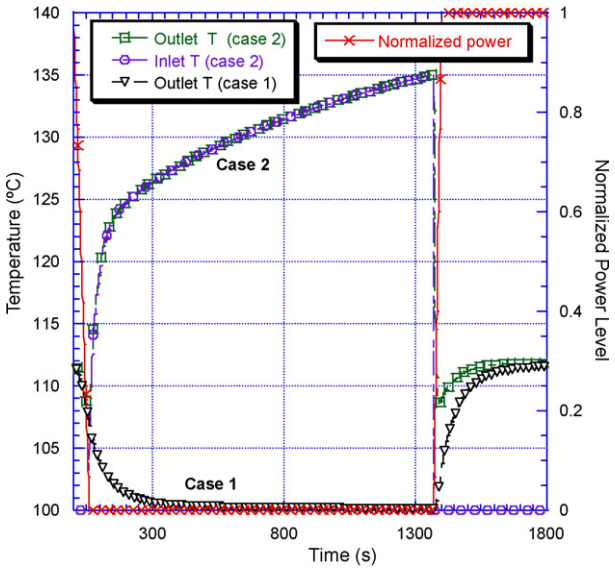

Fig. 8. The impact of heat exchanger operational scenarios on water temperature under an ITER inductive pulsed operation (Case 1 is a constant water inlet temperature of 100 °C during plasma off. Case 2 involves no heat exchanger during plasma off; water inlet temperature is 100 °C during plasma on.).

code, which allows for the development of a more sophisticated operational control algorithm in ISPC.

6. Summary and future work

The progress of ISPC development has been presented through a number of integrated simulations of key physical phenomena encountered in a fusion plasma chamber system. These analyses were all performed on CAD-based solid models. The ability to interface with a prototypical-sized CAD model offers realistic and consistent evaluations of design performance. The analysis has helped to identify areas where design improvement is needed. The combined effect on DCLL performance of liquid metal MHD, temperature gradients, and FCI structural deformation represents the most challenging task within this development. An example integrated thermo-fluid MHD simulation was performed for a single DCLL-like duct; however, much progress is still needed in this area. Interface engines were developed in order to pass the deformed surface data from the structural analysis to the thermo-fluid MHD analysis code for MHD velocity profile assessment. Complex velocity and temperature profiles calculated from a thermo-fluid code have been integrated into a mass transport equation in order to better account for the effects of temperature and flow on tritium permeation. A 3D neutronics analysis of a DCLL TBM integrated with the surrounding water-cooled frame has been performed, which reveals the importance of including geometric details into the performance prediction. Near-term ISPC development has been laid out, in which the main focus includes continuing to advance fusion-specific research codes such as HIMAG; enhancing and validating off-the-shelf third-party software for fusion plasma chamber applications; and developing integrated, multi-scale simulations.

Acknowledgements

This work was jointly supported by the U.S. Department of Energy Contract DE-FG03-ER52123, SBIR Phase I Award, and grants from Sandia National Laboratories as part of DOE's funded ITER project.

References

[1] M.E. Sawan, P.P.H. Wilson, T. Tautges, L.A. El-Guebaly, D.L. Henderson, T.D. Bohm, E.P. Marriott, B. Kiedrowski, B. Smith, A. Ibrahim, R. Slaybaugh, Application of CAD-neutronics coupling to geometrically complex fusion systems, Fusion Engineering (2009), SOFE 2009, 10.1109/FUSION.2009.5226505.
[2] A. Ying, M. Narula, M. Abdou, R. Munipalli, M. Ulrickson, P. Wilson, Toward an integrated simulation environment providing predictive capability for fusion plasma chamber systems, Fusion Science and Technology 56 (2) (2009) 918–924.
[3] J. Kotulski, et al., EM analysis update first 40 degree sector, Presented at SNL ITER Weekly Project Meeting.
[4] B. Smith, P.P.H. Wilson, M.E. Sawan, Three dimensional neutronics analysis of the ITER first wall/shield module 13, in: SOFE 2007. 2007 IEEE 22nd Symposium, Fusion Engineering (2007), 10.1109/FUSION.2007.4337910.
[5] A. Ying, M. Narula, H. Zhang, M. Abdou, Coupled transient thermo-fluid/thermal stress analysis approach in a VTBM setting, Fusion Engineering and Design 83 (10–12) (2008) 1807–1812.
[6] G. Aiello, F. Gabriel, L. Giancarli, G. Rampal, J.F. Salavy, Thermo-hydraulical and thermo-mechanical analysis of the HCLL-TBM breeding unit, Fusion Engineering and Design 83 (2008) 1227–1231.
[7] R. Munipalli, Multidisciplinary modeling for blanket physics, in: Presented at FNST Meeting, UCLA, August 18, 2009.
[8] S. Sharafat, DCLL TBM thermo-structural analysis status, in: Presented at FNST Meeting, UCLA, August 18, 2009.
[9] T. Tautges, P.P.H. Wilson, J. Kraftcheck, B. Smith, D. Henderson, Acceleration techniques for direct use of CAD-based geometries in Monte Carlo radiation transport, in: Proceedings of International Conference on Mathematics, Computational Methods & Reactor Physics (M&C 2009), Saratoga Springs, NY, May 3–7, 2009.
[10] T.J. Tautges, R. Meyers, K. Merkley, C. Stimpson, C. Ernst, MOAB: A Mesh-oriented Database, Sandia National Laboratories, 2004.
[11] T.J. Tautges, CGM: a geometry interface for mesh generation, analysis and other applications, Engineering with Computers 17 (2001) 299–314.
[12] K. Kizu, A. Pisarev, T. Tanabe, Co-permeation of deuterium and hydrogen through Pd, Journal of Nuclear Materials 289 (2001) 291–302.
[13] H. Zhang, A. Ying, M. A. Abdou, Integrated simulation on tritium permeation in the solid breeder unit, Fusion Eng. Des. (2010), doi:10.1016/j.fusengdes.2010.05.018.
[14] Z. An, A. Ying, M. Abdou, Application of discrete element method to study mechanical behaviors of ceramic breeder pebble beds, Fusion Engineering and Design 82 (2007) 2233–2238.
[15] Y. Gan, M. Kamlah, Discrete element modeling of pebble beds: with application to unaxial compression tests of ceramic breeder pebble beds, The Mechanics and Physics of Solids 58 (2) (February 2010) 129–144.
[16] N. Watanabe, M. Kubo, N. Yomoda, An 1D–3D integrating numerical simulation for engine cooling problem, SAE Technical Paper 2006-01-1603, in: 2006 SAE World Congress, Detroit, MI, April 3–6, 2006.