Available online at www.sciencedirect.com
ScienceDirect
Fusion Engineering and Design 82 (2007) 2217–2225
Fusion Engineering and Design
www.elsevier.com/locate/fusengdes

# Integrated thermo-fluid analysis towards helium flow path design for an ITER solid breeder blanket module

A. Ying [a,*], M. Narula [a], R. Hunt [a], M. Abdou [a], Y. Ando [b], I. Komada [b]

[a] *Fusion Engineering Sciences, Mechanical and Aerospace Eng. Department, University of California, Los Angeles, 420 Westwood Plaza, Los Angeles, CA 90095-1597, USA*
[b] *Software CRADLE Co. Ltd., Osaka 532-0011, Japan*

Received 31 July 2006; received in revised form 28 May 2007; accepted 29 May 2007
Available online 20 July 2007

**Abstract**

The successful design and development of a complex system, like the ITER test blanket module (TBM) warrants the need of extensive computer aided engineering (CAE) activities. In this light, a sophisticated numerical flow solver ('SC/Tetra' by CRADLE®), with a robust CAD interface, has been used to develop and evaluate helium coolant flow schemes for a solid breeder test blanket module design currently proposed by the US for testing in ITER. The traits of a particular cooling strategy for the TBM, namely the exit temperature of coolant, overall pressure drop, uniformity of temperature in the structure, robustness against transients, etc. can only be predicted by carrying out a complete three dimensional thermal-fluid analysis of the system in its entirety including all the structural and fluid components. The primary objective of this paper is to introduce the procedure for carrying out complex thermo-fluid analysis using the complete three dimensional CAD models of the TBM to evaluate the performance of TBM cooling schemes and to illustrate the way in which the results from these analyses can be useful towards a systematic design of an effective cooling solution for the test blanket module.


*Keywords:* Test blanket module (TBM); Helium coolant; Solid breeder; Thermo fluid analysis; ITER

## 1. Introduction

A sophisticated computational fluid dynamics (CFD) code with an ability to interface with three-dimensional CAD models has become an essential tool for practical engineering design. Such a tool can significantly reduce design uncertainties and help in the evaluation of different cooling schemes for the test blanket module (TBM), particularly in the designs where the flow distribution of the coolant is relatively complicated and involves many parallel flow paths. The TBM requires a robust design from the beginning, since no active control devices can be installed within the TBM to respond to changing operating conditions.

* Corresponding author. Tel.: +1 310 206 8815; fax: +1 310 825 2599.
*E-mail address:* ying@fusion.ucla.edu (A. Ying).

0920-3796/$ – see front matter 
doi:10.1016/j.fusengdes.2007.05.080

This calls in for an extensive analysis of the various proposed designs and cooling schemes for the TBM, under the entire range of operating conditions expected in ITER. The evaluation of a particular coolant distribution scheme for the TBM requires a complete three-dimensional thermo-fluid analysis with a coupled calculation of the flow field in the fluid part and the temperature field in both the fluid and solid components of the TBM.

The current US design for a solid breeder based TBM is the helium cooled ceramic breeder (HCCB) TBM. In the HCCB design, the breeding zones are housed behind a 'U' shaped first wall (FW) structural box. The top and bottom of the enclosure, housing the breeder unit, are closed by cap plates. The structural box is closed by a coolant manifold block located away from the FW that contains the coolant inlet and outlet supply and collection headers. The breeding zone is subdivided into breeder and beryllium beds, which are typically separated by cooling plates. The cooled structural box, with its internal cooling plates, forms the basic architecture of a ceramic breeder blanket design. Supplying all structures with adequate cooling is one of the most challenging tasks in HCCB blanket design. A previous design of the HCCB TBM, is provided in [1] to help the reader envision the complexity of the HCCB TBM system, the HCCB model presented in the present work reflects the most recent design iteration.

The proposed helium coolant flow circuit through the various components of the HCCB TBM is as follows; helium enters the TBM through an inlet pipe at 8 MPa and 573 K. It is then distributed into 16 parallel cooling paths for FW surface heat removal by means of inlet manifold, which has an inbuilt network of flow paths and buffer spaces for helium, to ensure a uniform coolant distribution. Each coolant path in the FW includes three passes, connected in series, which are expected to exert the same hydraulic resistance with each other under normal operating conditions. The flow leaves the first wall through a collecting manifold and is divided into two paths for cooling upper and lower caps and internal breeding zones. A double snake flow design is adopted to cool the breeding zone based on two flow paths. One path enters the breeding zone from the far left subunit and the other from the far right. Both streams flow radially to the front, make a turn and flow radially towards the back, to the collecting manifold. The flow is redistributed poloidally by means of poloidal manifold before injection into the parallel channels of the next breeder unit. Each stream is guided through the breeding coolant manifold towards the next breeder unit before merging into the outlet channel through the outlet manifold. The cooling circuit along with the associated manifolds is illustrated in Fig. 1.

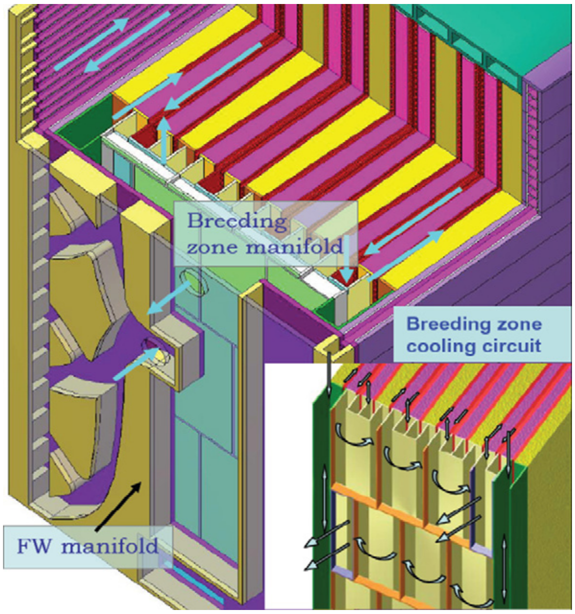


Fig. 1. Schematic view of FW and breeding zone coolant manifold designs (bottom half).

The main objective of the work described in this paper is to illustrate the procedure for carrying out a complete thermo-fluid analysis of helium cooling schemes, to evaluate their performance and impact on the TBM operation. An analysis of a similar nature for the European solid breeder blanket (HCPB) can be found in [2].

## 2. Thermo fluid analysis using SC/Tetra

### 2.1. The analysis methodology

The design analysis approach used in this study closely follows the well-established procedure of CAE for systematic product design. In the first step, a CAD model of the proposed design of the coolant distribution scheme in the TBM is developed using SolidWorks.

The complete CAD model of the TBM, including the solid and the fluid parts, is input to SC/Tetra, which provides the transient and steady state temperature field in the solid and fluid parts as well as the complete 3D flow distribution of the coolant in the manifolds and the coolant channels. The results from the thermo-fluid analysis can then be used to iterate the design to come up with the best coolant distribution scheme that ensures a uniform coolant distribution in all of the cooling channels and limits the maximum temperature in the system within the operational limits.

### 2.2. Introduction to SC/Tetra

SC/Tetra is a computational thermo-fluid analysis system based on an unstructured hybrid mesh finite volume method. The software is accompanied by a complete suite of analysis programs that handle the entire sequence of operations from reading the input geometry from the CAD file to the final post processing of flow field and temperature field results. The software suite includes a robust hybrid mesh generation program. The finite volume formulation used in the solver to discretize the governing equations is based on the cell vertex formulation. The flow analysis can be based on compressible or incompressible flow models that make use of segregated pressure based solution algorithms. The solver includes numerous turbulence models for modeling turbulent flows, the most common being the two equation k-$\varepsilon$ model and its variants. In addition the software allows definition and implementation of flow models developed by the user, making it possible to handle an extremely large variety of three dimensional thermal-fluid flow problems in complex systems. For a more detailed description of the software and the available models, the reader is referred to [3–5].

## 3. Thermo fluid analysis of TBM cooling schemes

Three different coolant flow path models are studied in this paper. The first two models include the helium coolant flow in the inlet manifold that feeds the coolant to 16 cooling channels in the first wall, followed by coolant discharge into the collection manifold. The two models studied here differ in the shape of the inlet and the collection manifold, which has a profound effect on the flow distribution in the 16 FW cooling channels. The third model includes the coolant flow in the breeder zone cooling channels in addition to the coolant flow in the first wall. It should be emphasized that all of these models present preliminary design of the proposed helium cooling circuit and will undergo several design iterations and associated flow analysis cycles before an optimal coolant flow configuration emerges. Apart from these models, a simplified version of the FW cooling channels model, with only three of the FW cooling channels and the associated inlet and collection manifolds, was used to carry out simple validation studies. The purpose of these validation studies was to identify the correct numerical procedure and proper mesh resolution to be used in the CFD model, so that the results obtained from the three-dimensional thermo-fluid analysis using SC/Tetra are meaningful.

### 3.1. The validation study

For a successful thermo-fluid analysis of the various helium flow configurations for the cooling of the TBM, it is important that correct numerical models be chosen from amongst those available in the SC/Tetra solver. Since the helium coolant used for TBM cooling is a gas, it can under go a significant change in density as it collects heat from the first wall and breeder zone, which warrants the use of a compressible flow model. Another important issue is the resolution of the computational mesh at the interface between the fluid and solid parts of the domain. In the SC/Tetra solver, while solving for turbulent flow, the logarithmic law of the wall is used to interpolate the velocity and temperature values at the nodes adjacent to the solid walls and it is important that the nodes adjacent to the walls are located at the proper distance for the log law to be valid. Hence, before a successful CFD analysis of the helium cooling circuit can be carried out, important choices have to be made regarding the use of the compressible versus the incompressible model, the correct mesh resolution adjacent to the wall, the correct turbulence model and the suitable set of boundary conditions. In order to help make these decisions, the three channel validation model described above was used and the results obtained by changing the various parameters and numerical models were compared with analytical correlations to decide on the best numerical procedure to be used.

In the three-channel validation model, the distribution of the flow field and temperature field was obtained in the inlet manifold, the FW cooling channels (three in number) and the collection manifold. The solid domain included the TBM FW ferritic steel structure as well as the 2 mm thick beryllium armor facing the plasma. A steady heat flux of 0.3 MW/m$^2$ was applied on the FW beryllium armor. For the purpose of analysis it was assumed that all the heat flux applied at the first wall was picked up by the helium coolant by using an adiabatic condition at the outer surface of the TBM structure exposed to the surroundings. The helium flow rate into the system was held constant at 0.06 kg/s at an inlet pressure of 8 MPa and an inlet temperature of 573 K. The RNG k-$\varepsilon$ model was used to obtain the turbulence viscosity and other turbulence exchange coefficients. The log law of the wall is valid until the non dimensional distance from the wall stays within the range $30 \leq y^+ \leq 1000$ (where $y^+$ refers to the standard non dimensional wall unit used in turbulent flow vernacular). In order to obtain correct results, it is important that the non-dimensional distance ($y^+$), corresponding to the first layer of computational nodes adjacent to the wall stays within the correct range for the log law to be valid. The three channel validation case was run with five different mesh resolutions at the wall namely, 0.1, 0.5, 1.0, 1.5 and 2.0 mm using an incompressible flow model. The results from these different cases were found to have a significant variation. The compressible flow model and the incompressible flow model were also compared at a wall mesh resolution of 1.0 mm to observe any significant effects of the variation of gas density to the overall flow distribution and heat transfer characteristics.

Table 1 shows a comparison between the computational results obtained with the incompressible flow model at five different wall mesh sizes. The important physical quantities of interest that have been obtained from the calculation and compared include the average heat transfer coefficient at the heat exchange surfaces of the FW cooling channels, the overall pressure drop in the system and the maximum temperature obtained in the FW structural material. A glance at the $y^+$ column of the table suggests that the very fine resolution of 0.1 mm clearly lies outside the region of validity of the log law of the wall and hence the results obtained for this case are incorrect. Of the remaining, the mesh resolution of 0.5 and 1.0 mm are well within the range of applicability of the log law while the coarser meshes border towards the upper limit. A simple calculation based on the Dittus-Boelter equation for turbulent heat transfer in pipe flows gives the value of average heat transfer coefficient on the cooling channel heat exchange surface to be 2779 W/m$^2$ K. The result from 3D thermo-fluid analysis using a mesh resolution of 1.0 mm differs from the above by 4%. This can be attributed to the fact that the Dittus-Boelter correlation is applicable in round pipes exposed to a uniform heat flux on the entire surface area. The Dittus-Boelter correlation provides us with a ballpark to check if the results obtained from SC/Tetra calculations are meaningful. The above study identifies that a wall mesh resolution that has an associated $100 \leq y^+ \leq 300$ should be used for analysis.

The compressible and the incompressible flow models give identical results for the average heat transfer coefficient on the FW cooling channel heat exchange surfaces, namely 2880 W/m$^2$ K. A conservative estimate of the overall pressure drop in the system can be obtained by the difference between the maximum and minimum pressure values in the system. Using this method, the overall pressure drop resulting from the compressible model was three times (0.09 MPa) compared to the incompressible model (0.03 MPa). This is

Table 1
Comparison of computational results at different wall mesh resolution

| Resolution (mm) | Average heat transfer coefficient (W/m$^2$ K) | $Y^+$ | Pressure drop (MPa) | Maximum temperature (beryllium, K) |
|---|---|---|---|---|
| 0.1 | 4200 | 5–30 | 0.023 | 755 |
| 0.5 | 3100 | 30–300 | 0.028 | 765 |
| 1.0 | 2900 | 100–300 | 0.030 | 762 |
| 1.5 | 2900 | 300–1000 | 0.032 | 761 |
| 2.0 | 2900 | 300–1000 | 0.037 | 760 |

reasonable as the density of the gas drops 11% due to expansion caused by heating in the compressible model which can cause a significant effect. The validation studies show that a compressible flow model with a near wall mesh resolution that places the wall adjacent nodes to satisfy $100 \leq y^{+} \leq 300$ for the log law to hold, should be the proper choice for thermo-fluid analysis set up with SC/Tetra to obtain satisfactory results.

### 3.2. First wall cooling analysis

The first wall of an ITER TBM will be subjected to an average surface heat flux of 0.3 MW/m$^2$. The RAFS structural material for the first wall has a maximum operating temperature of 823 K and the FW cooling scheme has to ensure that this constraint is met under all operating conditions. Another very important requirement on the FW cooling scheme is to avoid creation of local hot spots in the structural material that may result due to a non-uniform distribution of coolant in the FW cooling channels. The distribution of coolant in the FW cooling channels is dictated by the design of the inlet manifold that takes in high pressure (8 MPa, 573 K) helium coolant from the common supply line and distributes it amongst the 16 cooling channels. Though the uniform coolant distribution is the primary guiding principal behind the design for the inlet manifold, other factors like minimization of pressure drop, avoidance of regions with fluid recirculation etc. should also be kept in mind. Due to uncertainties in the surface thermal loading in ITER, it is desirable that the design of the inlet manifold allows for some degree of passive flow adjustment depending on the surface heating conditions. It is important to ensure that no choking points develop in the system and the flow can readjust to downstream heat loading conditions.

In this paper, two different inlet manifold designs have been analyzed for the FW cooling system. In the first design the incoming helium flow in divided into eight branches that feed into an open buffer to which the 16 channel inlets are connected. The thermo-fluid analysis model used to study the first manifold design with SC/Tetra included the inlet manifold, the FW cooling channels and the collection manifold with all the associated ferritic steel structure. The beryllium armor layer (2 mm thick) was added to the first wall and a constant heat flux of 0.3 MW/m$^2$ was applied to its surface. In the numerical model, the helium coolant was modeled as a compressible fluid, injected into the inlet manifold at an inlet pressure of 8 MPa and an inlet temperature of 573 K. The flow rate of helium in the system was kept fixed at 0.32 kg/s. The heat exchange surface between the beryllium layer and the ferritic steel structure was assumed to have no contact resistance. Turbulent heat transfer conditions were used at the heat exchange surface between the ferritic steel structure and the helium coolant by using the logarithmic law for temperature. As described in the section above, the mesh resolution at the wall was set at 1.0 mm, ensuring that the log law is applicable. For the purpose of fluid flow in the cooling channels, the pipes were assumed to be smooth by not taking into account any roughness factor at the walls.

In Fig. 2, the flow distribution in a cross section of the inlet and outlet manifold is shown. The cross section is cut at the middle of the manifold depth. The pattern

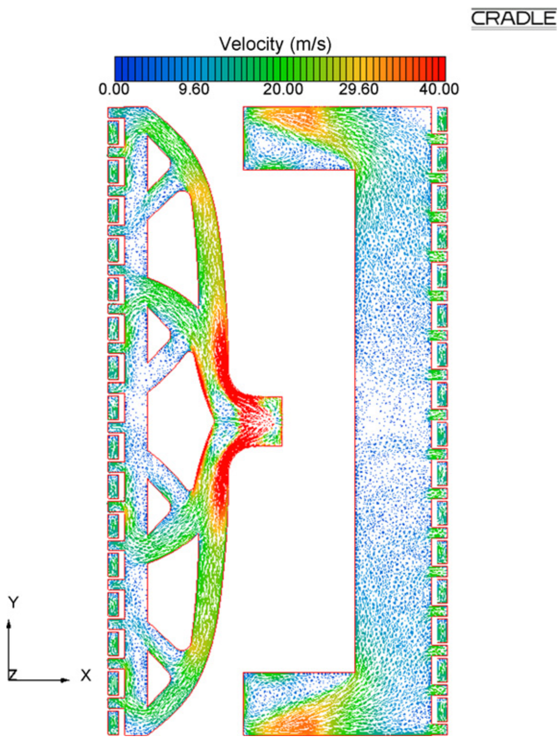


Fig. 2. Flow distribution in the inlet and collection manifolds of the first FW cooling scheme. The eight coolant distribution branches, feeding the buffer volume can be seen.

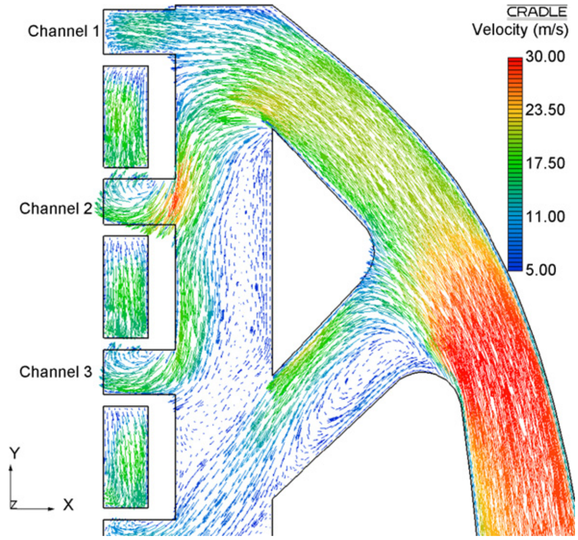


Fig. 3. Detailed flow distribution at the inlet of the first three cooling channels for the FW cooling. It can be observed that the flow enters the second and the third channel at an angle, causing a reduction in the flux.

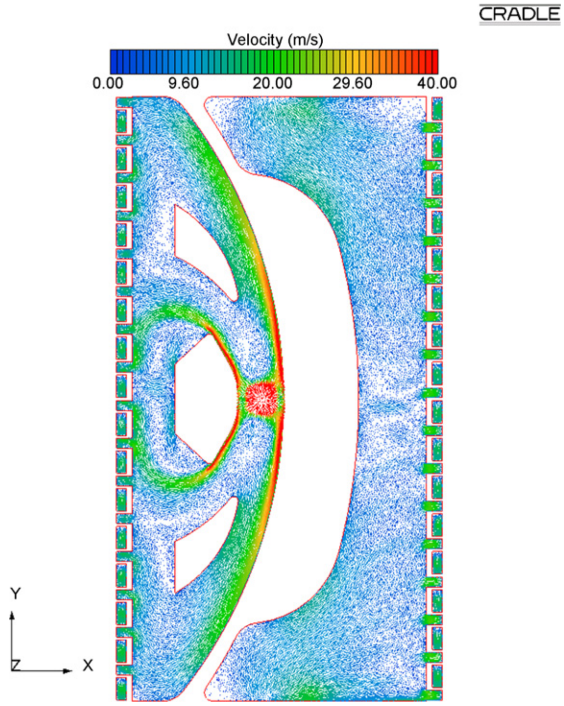


Fig. 4. Flow distribution in the inlet and the collection manifolds of the second FW cooling scheme. The branching in the inlet manifold has been reduced to four and the size of the buffer volume and the branch channels has been increased.

of velocity vectors show some interesting features of flow distribution in the inlet manifold and also hint at design changes that should be made to improve the flow behavior. As the coolant enters the inlet manifold, it is immediately divided into two streams which flow in opposite directions so as to help in filling the buffer volume uniformly. The flow splitting at the inlet is not effective because of the creation of recirculation zones. The two streams at the inlet are further split into four branches. During branching, the fluid has to turn by an obtuse angle to enter one of the legs and this causes fluid recirculation and an uneven distribution of flow in the two legs of a branch. The legs, in which the fluid has to turn by an obtuse angle relative to its original flow path, should be avoided. In Fig. 3, the flow pattern close to the inlet section of the first three cooling channel pipes is shown, it can be observed that in the first channel the flow vectors are normal to the inlet area while in the second and third channel, the flow vectors enter at an angle to the inlet area, leading to a reduction in the inlet mass flux and creation of recirculation within the channel.

The design of the inlet and outlet manifolds was modified in the second model with a view to simplify its construction and enhance its flow distribution properties. In the second design, the incoming helium flow is divided into four branches that feed into the buffer volume preceding the FW cooling channel inlets. The volume of the buffer zone has been doubled as compared to the earlier design. The size of the branching channels has also been increased. The analysis model includes the inlet manifold, the FW cooling channels and the collection manifold with all the associated ferritic steel structural material and the FW beryllium armor. The numerical models used for the analysis of the second model were identical to those used for the first. In Fig. 4 the distribution of the flow in the second design of the inlet and outlet manifolds is shown by cutting a cross section through the middle of the manifold depth. Fig. 5 shows a comparison of the coolant distribution obtained in all 16 FW channels in the first and second design. Even though, the second design is an improvement over the first, the three channels in the middle of the first wall remain underfed and

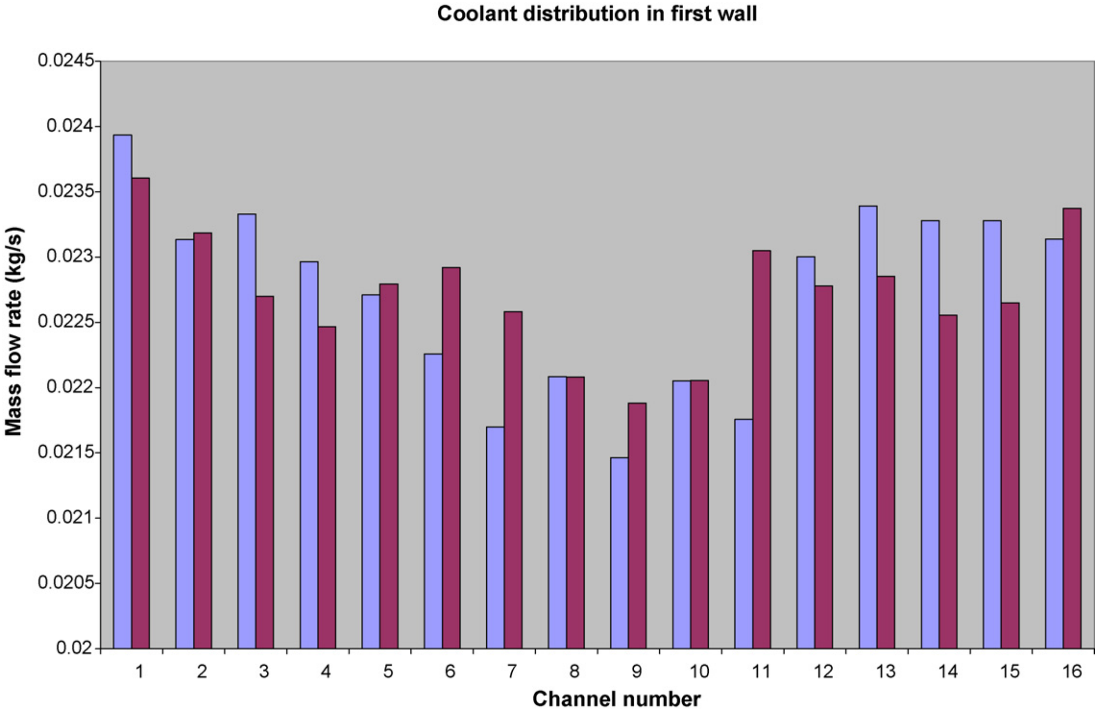


Fig. 5. A comparison of the coolant flow rate obtained in the 16 FW cooling channels with the two FW cooling schemes. The bars on the left indicate the first cooling scheme and those on the right represent the second.

require some further modifications to the inlet manifold design.

An interesting feature can be observed regarding the temperature distribution on the first wall structure under both cooling schemes. A temperature hot spot is formed in the first wall structure at the bottom of the TBM due to the difference in the pattern of heat transfer next to the boundaries as opposed to the interior. However, the hot spot is not observed at the top boundary because in both the FW cooling schemes, the cold leg of the three pass FW channel faces the top boundary and the hot leg of the three pass channel faces the boundary at the bottom of the TBM. Adiabatic conditions on the TBM structure surface causes the entire heat flow from the structure to the coolant. Since heat transfer is deteriorated due to presence of hot leg close to the bottom boundary, a hot spot is formed at the bottom. In Fig. 6, the variation of the temperature on the FW structure exposed to the uniform surface heat flux of 0.3 MW/m$^2$ is presented along a vertical line at the center of the first wall. The hot spot can be observed at the bottom. The cyclic high and low temperature pat-

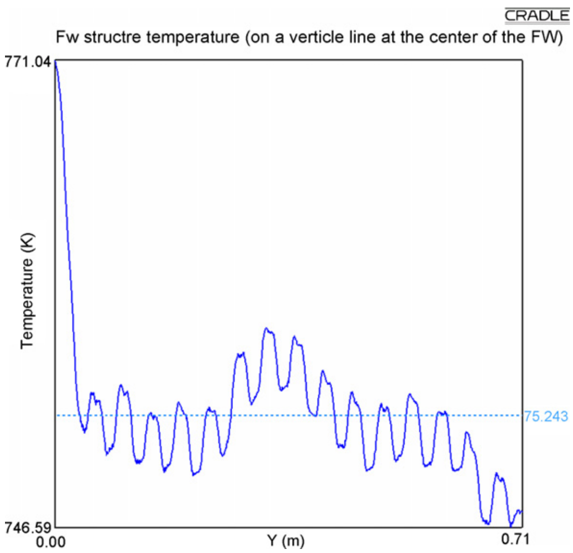


Fig. 6. Temperature distribution in the first wall structure, drawn along a vertical line running at the center of the first wall. This profile is obtained with the second cooling scheme. The first cooling scheme results in a similar profile.

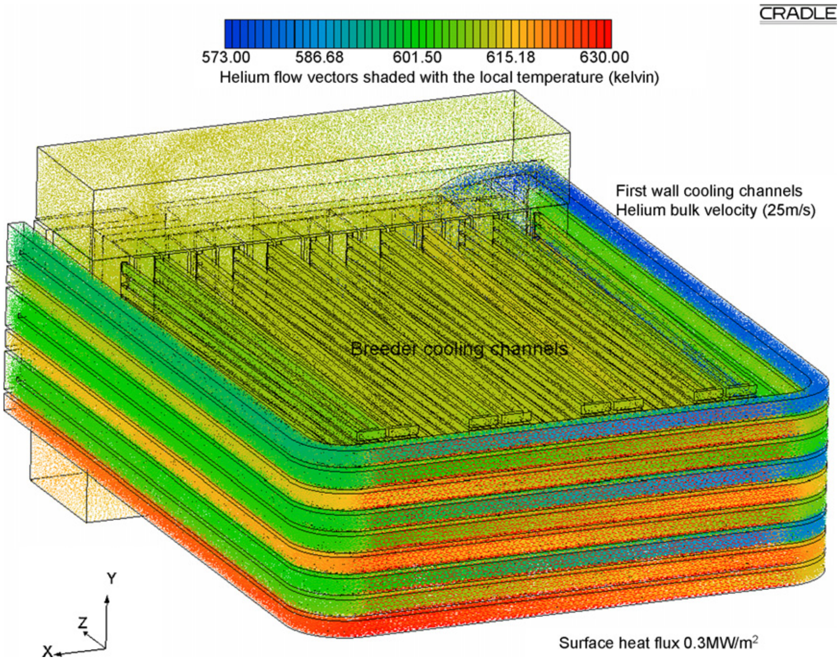


Fig. 7. Variation of helium coolant temperature in the entire flow circuit, including the FW cooling channels and the breeder zone.

tern on the FW structure corresponds to the hot and cold legs of the three pass cooling channels. The FW hot spot as observed in the current cooling schemes needs to be eliminated and one possible solution is to reverse the flow path for helium in the bottom half of the FW channels (so that the cold leg faces the boundary as at the top).

### *3.3. Helium flow in breeder cooling channels*

In this model, the helium flow circuit under study included the FW cooling channels followed by coolant flow in 16 parallel channels in the breeder zone. The model used had roughly one sixth of the poloidal extent of the actual TBM. All the numerical modeling parameters for this model were kept the same as in the analyses described above. The primary goal of this simulation was to establish that the software can handle extremely complex flow geometry involving numerous channels and distribution manifolds. In Fig. 7, the temperature distribution of helium coolant is shown in the entire flow circuit. A detailed thermo fluid analysis of the helium coolant flow in the entire TBM and its role in the design of breeder channel flow distribution manifolds will form the subject of the next paper.

## 4. Conclusion and future work

In this paper, a procedure for carrying out a complete thermo fluid analysis, for evaluation of the helium cooling schemes for TBM has been described. The emphasis has been on the analysis of two different cooling schemes for the FW. The analysis has helped to identify problems like a non uniform coolant distribution in the FW cooling channels and creation of hot spot on the FW structure due to the configuration of the FW cooling channels, which will be removed in the future design iterations. A detailed thermo-fluid analysis of the entire helium flow circuit in the HCCB TBM will also be carried out in the future, including helium flow in the first wall and the breeding zone cooling plates. It is possible to export the temperature field and the computational mesh obtained from SC/Tetra in the solid domain to a finite element structural analysis system for a coupled thermo-fluid thermal stress analysis of the TBM.

## Acknowledgement

This work was supported in part by the US Department of Energy Office of Fusion Energy Science under DE-FC02-04ER54698.

## References

[1] A. Ying, M. Abdou, P. Calderoni, S. Sharafat, M. Youssef, J. An, A. Abou-Sena, E. Kim, S. Reyes, S. Willms, R. Kurtz, Solid breeder test blanket module design and analysis, Fusion Eng. Des. 81 (2006) 659–664.

[2] M. Ilic, B. Dolensky, L.V. Boccaccini, R. Meyder, C. Polixa, P. Schanz, Thermo-Hydraulics and Mechanical Design of the First Wall of the EUROPIAN HCPB Test Blanket Module, Presented at 24th SOFT, 11–15 September 2006, Palace of Culture and Science, Warsaw, Poland.

[3] Solver Reference of User's Guide of SC/Tetra Version 5.0.

[4] T. Matsushima, An automatic mesh generator based CFD system to be used as a Design Tool, SAE paper 2001-01-0037, 2001.

[5] Q. Fan, CFD simulation of pressure drop in line pipe, SAE paper 2006-01-1443, 2006.