Fusion Engineering and Design 86 (2011) 1997–2002

Contents lists available at ScienceDirect

Fusion Engineering and Design

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

Thermo-hydraulic and structural analysis for finger-based concept of ITER blanket first wall

Byoung-Yoon Kim*, Hee-Jae Ahn

ITER Korea, National Fusion Research Institute, Daejeon, 305-333, Republic of Korea

ARTICLE INFO

Article history:
Available online 8 May 2011

Keywords:
ITER
Blanket
First wall
High heat flux
Finite element method

ABSTRACT

The blanket first wall is one of the main plasma facing components in ITER tokamak. The finger-typed first wall was proposed through the current design progress by ITER organization. In this concept, each first wall module is composed of a beam and twenty fingers. The main function of the first wall is to remove efficiently the high heat flux loading from the fusion plasma during its operation. Therefore, the thermal and structural performance should be investigated for the proposed finger-based design concept of first wall. The various case studies were performed for a unit finger model considering different loading conditions. The finite element model was made for a half of a module using symmetric boundary conditions to reduce the computational effort. The thermo-hydraulic analysis was performed to obtain the pressure drop and temperature profiles. Then the structural analysis was carried out using the maximum temperature distribution obtained in thermo-hydraulic analysis. Finally, the transient thermo-hydraulic analysis was performed for the generic first wall module to obtain the temperature evolution history considering cyclic heat flux loading with nuclear heating. After that, the thermo-mechanical analysis was performed at the time step when the maximum temperature gradient was occurred. Also, the stress analysis was performed for the component with a finger and a beam to check the residual stress of the component after thermal shrinkage assembly.

1. Introduction

The ITER blanket first wall is one of the core components for the development of the future fusion plant. The blanket is located in the vacuum vessel in order to protect them against the cyclic thermal load from high heat flux during the fusion operation. The blanket covers a surface of 680 m^2 around plasma and the total number of blanket modules is 440. Each blanket consists of a first wall and a shield block. Currently, the conceptual design of the finger-typed first wall has been extensively studied. The finger-typed first wall module is composed of a supporting beam with twenty fingers. Each finger consists of beryllium tiles, cooling pipes, CuCrZr (Cu alloy) and a stainless block (SS block) [1–4].

The main function of the first wall is to remove efficiently the high heat flux loading from the fusion plasma during its operation. Therefore, the thermal and structural performance should be investigated for the proposed finger-based design concept of first wall. In this work, thermo-hydraulic and structural analyses were performed considering different surface heat loads and nuclear heating condition using CFX and ANSYS. Firstly, the static thermo-hydraulic analysis was performed to obtain the pressure drop and temperature profiles for a unit finger model. Then the structural analysis was carried out using the maximum temperature distribution obtained in thermo-hydraulic analysis. Finally, the transient thermo-hydraulic analysis was performed for the generic first wall module to obtain the temperature evolution history considering a moderate heat flux loading and nuclear heating. After that, the thermo-mechanical analysis was performed at the time step when the maximum temperature gradient was occurred. Also, the stress analysis was performed for the component with a finger and a beam to check the residual stress of the component after thermal shrinkage assembly.

2. Finite element models

2.1. A model description

In this work, a unit finger model and a full module were introduced for the analyses, respectively, as shown in Fig. 1. The length, the width and height of a unit finger are 671 mm, 97 mm and 111 mm. A unit finger consists of twenty eight Be tiles, two Cu alloy blocks, four cooling pipes and a SS block. A full module includes twenty fingers and a supporting beam. The size and weight of a module are 1.4 m^2 and 1.2 ton, respectively. For the analysis, a half

* Corresponding author. Tel.: +82 42 879 5646; fax: +82 42 879 5699.
E-mail address: bykim@nfri.re.kr (B.-Y. Kim).

0920-3796/$ – see front matter © 2011 Elsevier B.V. All rights reserved.
doi:10.1016/j.fusengdes.2011.04.006

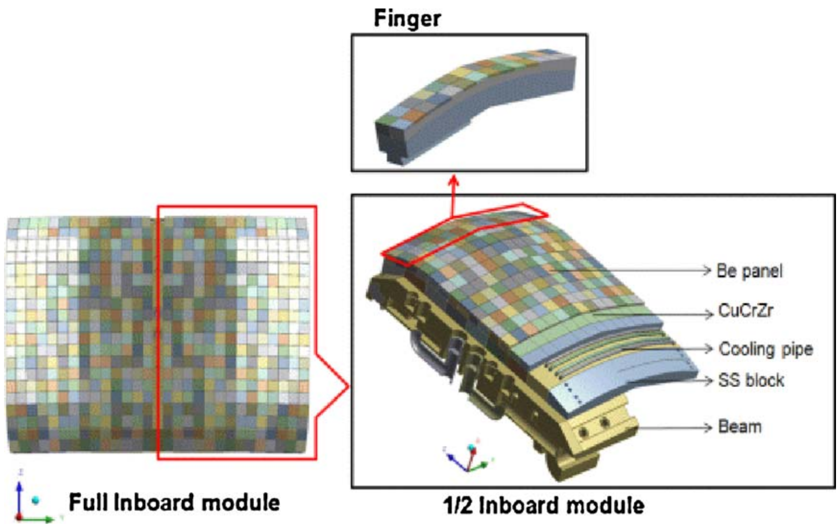

Fig. 1. A unit finger and a first wall module for a typical finger-typed blanket first wall.

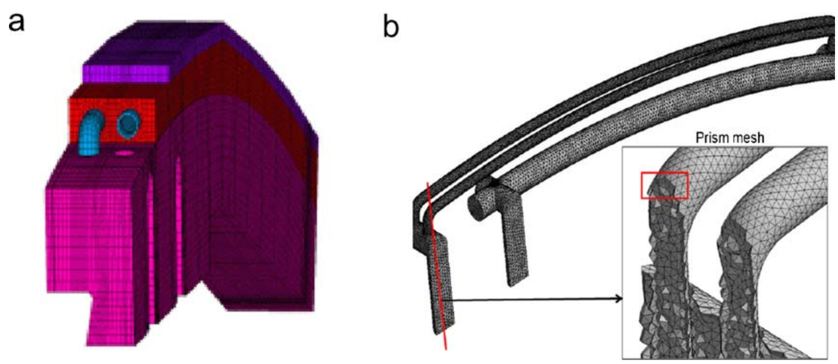

Fig. 2. Finite meshes of a unit finger: (a) a solid model for structural analyses and (b) a fluid model for hydraulic analyses.

of models for a unit finger and a module were used due to their symmetric system, in order to improve computational efficiency.

2.2. A unit finger

Element types of structural analyses were composed of mainly hexa-elements except Cu alloy due to its geometrical complexity. Average element size was set to be a 5 mm. The total numbers of element and nodes for a solid model were 122,189 and 594,985, respectively. As a result of shape testing, there was no meaningful error in this finite element model. Regarding thermal and hydraulic analysis, total numbers of elements and nodes for a fluid model were 410,000 and 130,000, respectively, and the average cell size was 3 mm. The finite meshes for a unit finger were shown in Fig. 2.

2.3. A module

For a module, element types of structural analyses were composed of mainly tetra-elements. Average element size was set to be a 10 mm. The total numbers of element and nodes for a solid model were 2,141,631 and 444,904, respectively. Regarding thermal and hydraulic analysis, total numbers of elements and nodes for a fluid model were 7,470,000 and 2,490,000, respectively. The average cell size was 8 mm, although average element size of interfaces with solid area was 0.3 mm. The finite meshes for a module were shown in Fig. 3. For a shrinkage fit assembly simulation by temperature

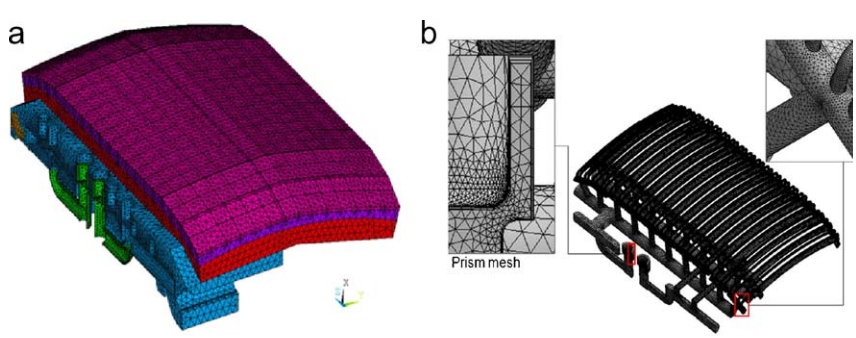

Fig. 3. Finite meshes of a typical first wall module for (a) a solid model and (b) a fluid model.

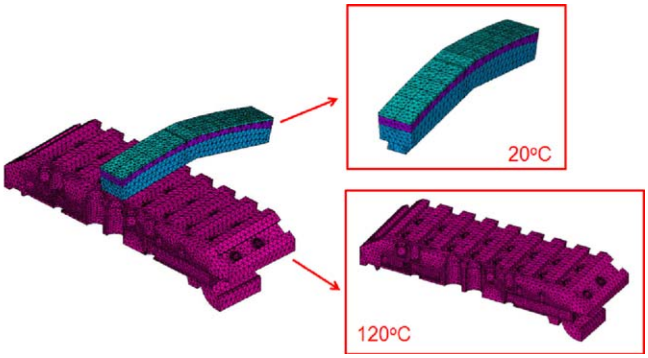

Fig. 4. Finite meshes for a shrinkage fit assembly simulation.

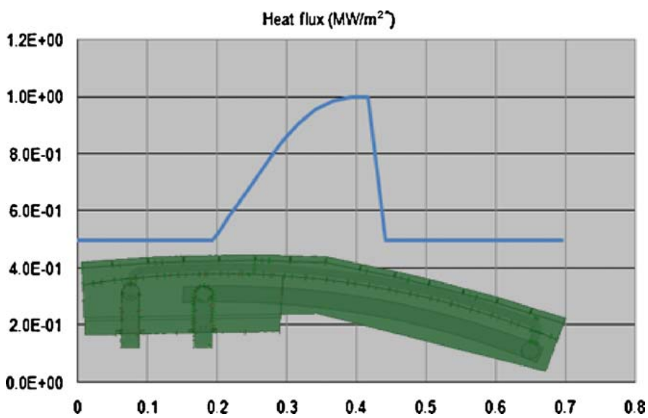

Fig. 5. Variable heat flux for a unit finger (F-3).

gradient, a unit finger and a beam structure was modeled as shown in Fig. 4.

3. Loading conditions

In this work, five cases were investigated as shown in Table 1. Three cases were investigated using static thermo-hydraulic and structural analyses for a unit finger. A case study was carried out using the transient condition for a module. The last case study was performed to simulate shrinkage fit assembly. Regarding a finger model, constant and variable heat flux loading were considered. Constant heat flux loading was applied for the case F-1 and F-2, while in case F-3, as shown in Fig. 5, heat flux variable along the unit finger was applied. The volumetric nuclear loading for Be, CuCrZr and SS were, respectively, 3.73×10^6 MW/m^3, $6.079 \times 10^6\,e^{-3.51x}$ MW/m^3, $6.315 \times 10^6\,e^{-6.59x}$ MW/m^3, where x is the distance from the surface of first wall. The transient analysis (case M-1) was carried out considering five cycles (400 s on and 1400 s off). Heat flux loading and nuclear loading were applied during operation time in the transient analysis. For simulating the shrinkage fit of a finger in a beam, the steady structural analysis was performed. The temperature gradient was set to 100 °C (120 °C in a supporting beam and 20 °C in a finger, respectively).

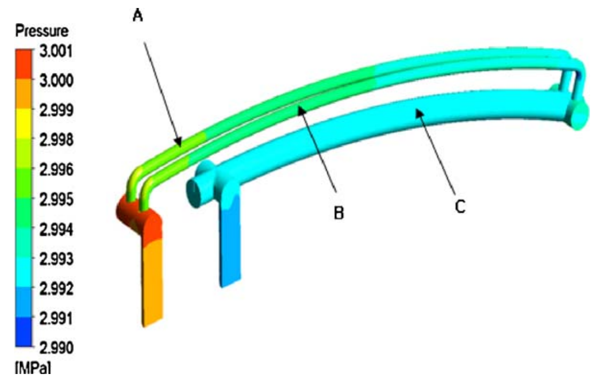

Fig. 6. Profiles of pressure at different locations for a unit finger.

Table 2
Pressure, velocity and HTC for a unit finger .at different locations (A–C indicate locations in Fig. 6).

	A	B	C
Pressure (MPa)	2.995	2.994	2.992
Velocity (m/s)	2.09	2.01	0.728
HTC (W/m^2 °C)	11398	10352	3863

Table 3
Maximum temperature and Von-Mises stress values of each component for cases F-1, F-2, and F-3.

	Maximum	F-1	F-2	F-3	Remark
Be tiles	Temp (°C)	385	678	516	Fig. 7(a)
	V.M.S. (MPa)	796	1740	906	–
CuCrZr	Temp (°C)	353	598	481	–
	V.M.S. (MPa)	1190	2560	1180	–
Pipe	Temp (°C)	244	381	315	–
	V.M.S. (MPa)	625	1310	1350	
SS block	Temp (°C)	335	561	493	Fig. 7(b)
	V.M.S. (MPa)	901	1763	1290	Fig. 8

4. Analysis results

4.1. Results for a unit finger

The static thermo-hydraulic analysis was performed for a unit finger. The pressure profile at different locations was shown in Fig. 6. The total pressure drop of a unit finger was obtained as 8910 Pa. Table 2 represents the values of pressure, velocity, and heat transfer coefficient (HTC) for a unit finger at different locations represented in Fig. 6.

The maximum temperature and stress values were shown in Table 3. According to increase of heat flux loading and existence of nuclear loading, the maximum temperature and stress values were increased in the all components.

The temperature and stress profiles for case F-3 were shown in Figs. 7 and 8, respectively. From the temperature profile in Fig. 7, temperature in most area was relatively acceptable except the left

Table 1
Loading combinations for a finger and a module.

Case	Heat flux load	Nuclear load	Fluid velocity	Water pressure	Water temp.
F-1	0.5 MW/m^2	X	1.4 m/s (0.312 kg/s)	3 MPa	100 °C
F-2	1 MW/m^2	X			
F-3	0.5–1 MW/m^2	O			
M-1	0.2 MW/m^2	O	3.45 kg/s		
M-2	Assembly simulation between a finger and a beam				

F and M represent a finger and a module, respectively.

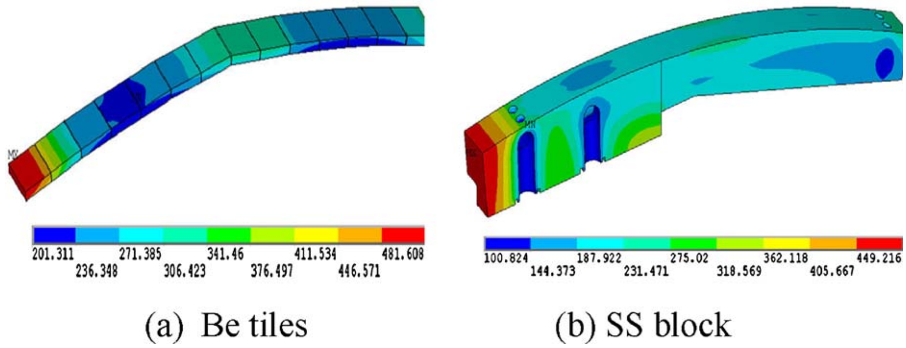

Fig. 7. Temperature distribution for case F-3.

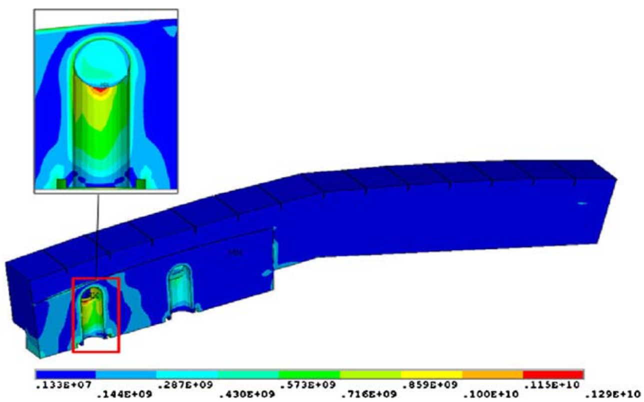

Fig. 8. Von-Mises stress distribution for case F-3.

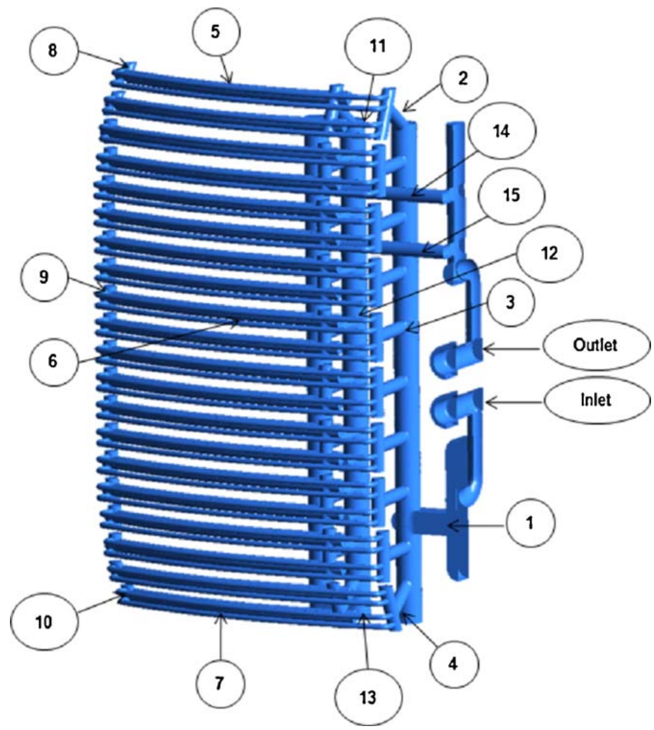

Fig. 9. Checkpoint of fluid.

part of components. This is due to the lack of the cooling capacity in the left part of the components. Therefore, the layout of cooling pipes is necessary to be extended to the left side of finger to improve the cooling efficiency. In Fig. 8 the von-Mises stress distribution was shown for case F-3. Due to the temperature mismatch and the material difference, the von-Mises stress was highly localized in tube end region. This stress localization could be mitigated according to the extension of cooling layout.

In cases of unit finger, the parametric study was done to check the impact on temperature and stress according to different heat flux loading and the existence of nuclear loading. In reality, the unit finger or first wall is subjected to cyclic heat flux loading due to the pulsed plasma operation. Therefore the static analysis could overestimate the values of temperature and stress, if the transient loading does not reach equilibrium state. Therefore, the maximum values of temperature as well as stress would be much reduced if the cyclic transient loading is introduced. Nevertheless, the cooling layout should be modified to improve the cooling efficiency.

Table 4
Values of pressure, mass flow rate, temperature and HTC at different locations in a first wall module.

	Pressure (MPa)	Mass flow rate (kg/s)	Temp. (°C)	HTC (W/m^2 K)	Remark
Inlet	3.1792	6.9	100.00	19747	
1	3.1236	3.45	100.16	41064	Delivery channel inlet
2	3.1251	0.0013	101.84	5839	Vertical riser top
3	3.1248	0.1634	100.39	10285	Vertical riser middle
4	3.1257	0.8590	100.61	17548	Vertical riser bottom
5	3.1234	0.0774	113.48	6122	Finger tube in steel top
6	3.1227	0.0779	113.73	6163	Finger tube in steel middle
7	3.1210	0.1326	107.48	9255	Finger tube in steel bottom
8	3.1230	0.0093	117.69	9616	Cooling pipe inlet top
9	3.1224	0.1602	122.01	11282	Cooling pipe inlet middle
10	3.1200	0.0192	110.95	26208	Cooling pipe inlet bottom
11	3.1224	0.0640	123.25	5899	Cooling pipe outlet top
12	3.1216	0.0249	122.94	7333	Cooling pipe outlet middle
13	3.1180	0.1258	131.62	11233	Cooling pipe outlet bottom
14	3.0995	1.7103	124.04	17714	Delivery channel outlet top
15	3.0909	1.7377	124.25	19003	Delivery channel outlet bottom
Outlet	3	6.9	124.34	57154	

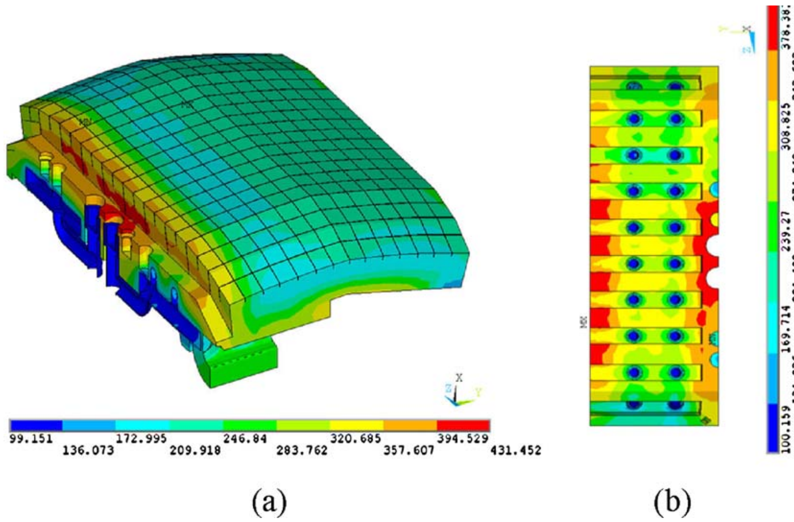

Fig. 10. Temperature distribution for case M-1.

4.2. Results for a module

The transient thermo-hydraulic analysis was performed for a module. The temperature rise from the thermal loading was 24 °C. Values of pressure, mass flow rate, temperature and HTC at different locations (Fig. 9) in a first wall module were shown in Table 4. The total pressure drop between inlet and outlet of a module was obtained as 0.18 MPa. The majority of pressure loss (0.15 MPa, 83% of total pressure drop) was generated in inlet and outlet areas due to their geometrical complexities. Resulting temperatures in a module and a beam were shown in Fig. 10. The temperature values were in the reasonable level, although the high temperature gradient was generated in the middle area, i.e., a supporting beam structure and some part of fingers.

The structural analysis was done at the timestep which has the maximum temperature gradient in solid area. The von-Mises stress profiles were shown in Fig. 11. Thermal stresses in a module were generally acceptable in the most area, although there were some high stress areas in cooling pipe, SS block and a beam structure. The maximum von-Mises stresses of Be, CuCrZr, cooling pipe, SS block and a supporting beam were 209, 342, 660, 829, and 1350 MPa. If the design including cooling layout of finger and beam is modified,

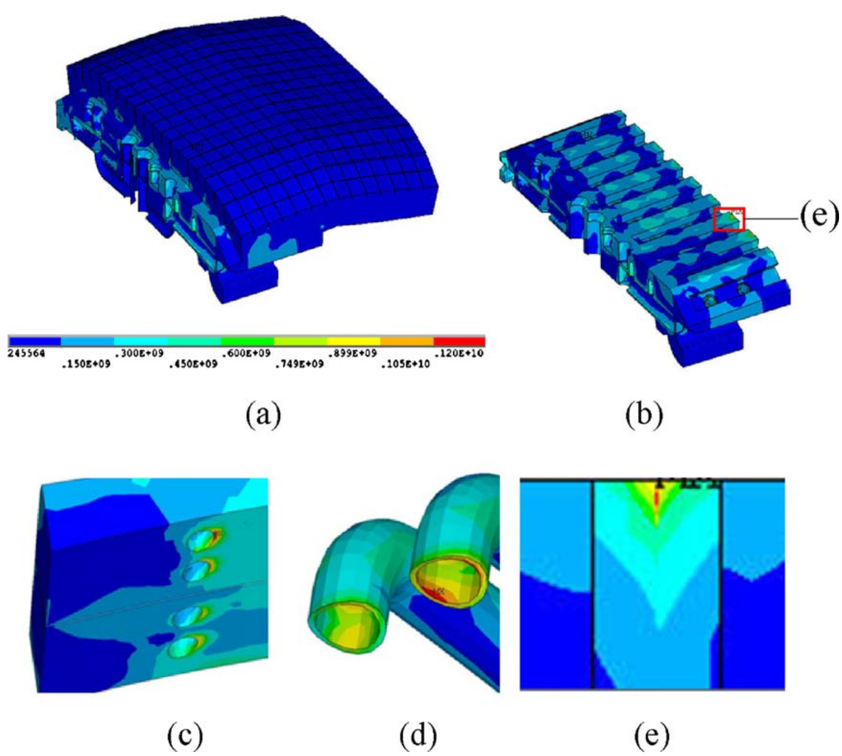

Fig. 11. Von-Mises stress distribution profiles in (a) a module, (b) a supporting beam, (c) SS blocks in finger, (d) cooling tube in finger, and (e) a magnifying view of red square in (b).

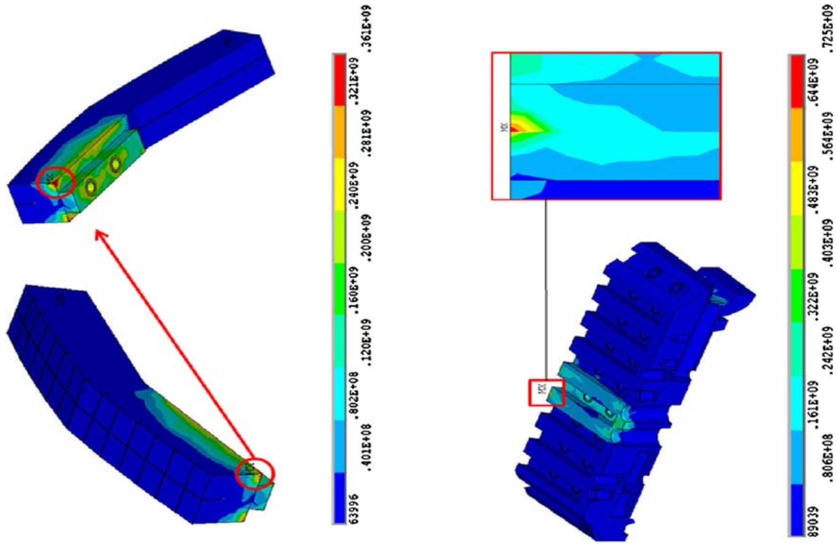

Fig. 12. Temperature distribution for case M-1.

temperature gradient could become sufficiently lower than the current result. Then, the locally high thermal stress of a module would be avoidable.

4.3. Results for an assembly simulation

The structural analysis was performed to check the residual stress of the component after thermal shrinkage assembly. The stress profiles were shown in Fig. 12, and the maximum values in a finger and a beam were 361 MPa and 726 MPa, respectively. However, the stress localization would be relieved in the reality due to slipping in the interface area.

5. Conclusion

From the hydraulic and structural analysis of a unit finger and a module, the temperature and the stress results were obtained for various thermal loading cases and an assembly case. From the analysis, the stress and temperature were mostly in reasonable range. However, there were locally high temperature and stress areas in this system. Since the design activity of blanket first wall has been on-going, they could be solved through design modification.

Acknowledgments

This research was supported by National R&D Program through the National Research Foundation of Korea (NRF) funded by the Ministry of Education, Science and Technology & Ministry of Knowledge Economy.

References

[1] A.R. Raffray, et al., Fus. Eng. Des. 93 (85) (2010).
[2] D.L. Youchison, et al., Fus. Eng. Des. 83 (2008) 1025.
[3] D.W. Lee, et al., Thin Solid Films (2010) (available online).
[4] A. Ying, et al., Fus. Eng. Des. (2010) (available online).