§18. Numerical Simulation of Heat Load Response of Plasma Facing Component

Nogami, S., Hasegawa, A. (Tohoku Univ.),
Tamura, H.

1. Introduction

Evaluation and prediction of heat load response in plasma facing components such as divertor and blanket first wall during operation is demanded to keep structural reliability of them. Heat load test using a component specimen is one of the promising and available methods for evaluation of heat load response of them. However, it is difficult to clarify the damage accumulation and fracture mechanism when using such experimental method. Therefore utilization of the numerical simulation methods in addition to the experimental ones is important because evaluations under various conditions can be performed.

2. Heat load test for VPS-W coated F82H

Vacuum plasma spraying (VPS) method is one of the promising engineering coating methods for tungsten (W) on the structural materials such as F82H steel. Since the operation temperature of VPS-W coating should be limited below the maximum utilization temperature of F82H (~873 K), it is possible that the material properties related to the heat load response such as the density, strength and thermal conductivity are different from those of the bulk W materials. Therefore, evaluation of the heat load response is essential for the VPS-W.

Tokunaga *et al.* reported the heat load test for the VPS-W coated F82H using ACT facility of NIFS [1]. As the result of this test, there observed no cracks after 100 cycles of pulsed heat loads at 4.8 MW/m^2, where the W surface temperature was 1090 K (coolant temp.: 384 K). For the cyclic heat load at 5.5 W/m^2, in contrast, the surface temperature increased up to 1150 K (coolant temp.: 414 K) and cracks were formed remarkably after only 16 cycles.

3. Objective

This study focused on the heat load test by Tokunaga *et al.*. By simulating this heat load test using a finite element analysis (FEA), crack initiation process and condition of the VPS-W during the heat load was investigated.

4. Results and discussion

As the result of the FE calculation, the damage accumulation and crack initiation processes during the heat load test was assumed as follow:

(1) During temperature rising term due to heat load, the heat-loaded area was thermally expanded. On the other hand, the non-heat-loaded area did not expand in comparison with that. As a result, compressive stress was generated in the heat-loaded area.
(2) If the compressive stress of the heat-loaded area reached the yield stress of the VPS-W, plastic deformation occurred in this area.
(3) During the cooling term after stopping the heat load, the temperature of the VPS-W was decreased and thermally shrunk. The shrinkage of the heat-loaded area was relatively small compared to the non-heat-loaded area because the plastic deformation occurred in the heat-loaded area during the temperature rising term .
(4) As a result, if the tensile stress, which exceeded the strength of the VPS-W, was generated in the heat-loaded area, the crack initiated in this area.

In general, the temperature distribution (see Fig. 1) and stress change behavior in the VPS-W was successfully simulated by the FEA in this study. On the other hand, stress value was considered to be slightly different. This difference was attributed to the difference in the material property values between the VPS-W and bulk W. Most of the material property values of the FEA in this study were those of the bulk pure W in the open literatures.

In addition, thermally occurred residual stress during the VPS coating process is one of the factors which will change the calculation accuracy. The residual stress was considered to be above 1 GPa by the 1 dimension elastic mechanistic calculation.

Based on this research, understanding of the accurate material property values and the residual stress generated during the VPS coating process is the key to increase the simulation accuracy of the heat load response of the plasma facing component.

1) T. Tokunaga, et al., J. Nucl. Mater., (2012).

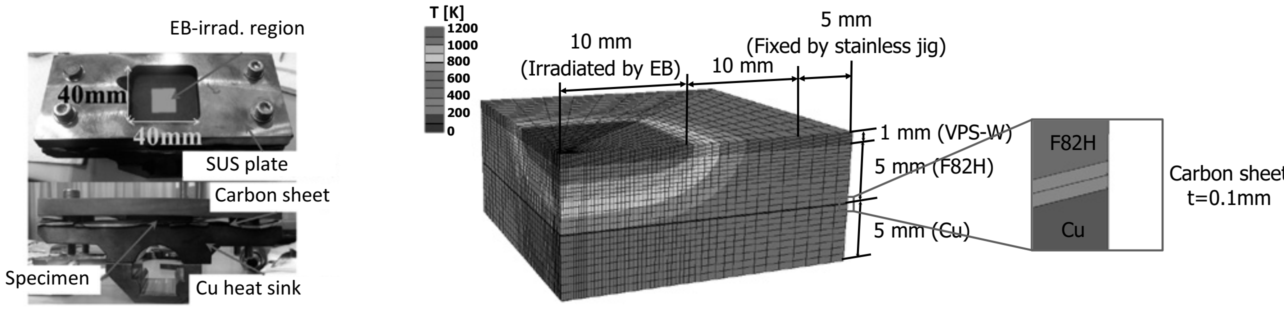

Fig. 1. Test jig appearance and typical temperature distribution calculated by FEA on ACT electron beam heat load test (surface temperature of VPS-W: 1150 K, coolant temperature: 500 K)