Thermo Mechanical Analysis of Divertor Test Mock-up Using COMSOL Multiphysics

Y. Patil[1], D. Krishnan[1], S. S. Khirwadkar[1]

[1]Institute for plasma research, Bhat, Gandhinagar, Gujarat, India

Abstract

Divertor is act as an exhaust for the nuclear fusion reactor. Main function of a divertor is to remove the heat flux from the plasma. Plasma facing components of the divertor are made up of Carbon (Graphite/CFC) and tungsten like materials[1]. Hence these materials are exposed to the transient heat loads up to 10MW/m^2. Thermo mechanical behavior of Graphite test mock-up under the transient heat loads was simulated using heat transfer and structural mechanics modules of COMSOL Multiphysics[2]. 3D semi-model of Graphite test mock-up was developed in COMSOL Multiphysics. Material properties and boundary conditions are assigned to it. Extracted heat flux by mock-up is 7.22 MW/m^2 which corresponds to incident heat flux 10MW/m^2 was incident on mock-up and temperature profile has been calculated as shown in figure 1. Stress developed during transient heat loads at various parts of mock-up has been calculated using structural mechanics[3].

Reference

[1] J Link, Plasma facing materials and components for future fusion devices development, characterization and performance under fusion specific loading conditions, Phys. Scr. T123 (2006) 45-53
[2] Comsol multiphysics user manual
[3] Luděk Pešek, Petr Šulc, FE model of thermo-mechanical interaction in rubber block under dynamic cyclic, Proceedings of the COMSOL Users Conference 2006 Prague
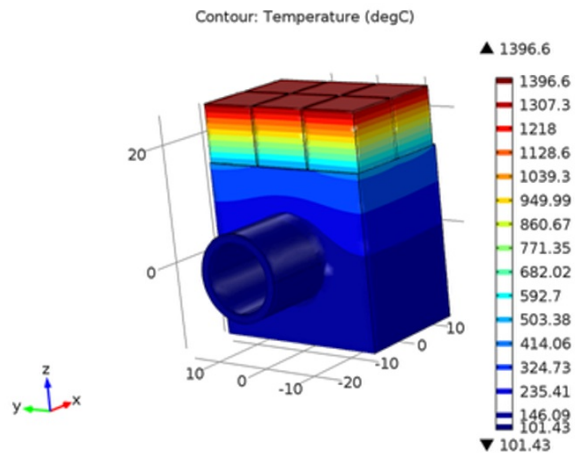
Figures used in the abstract

Contour: Temperature (degC)

Figure 1