Thermo-mechanical Analysis of Divertor test mock-up using Comsol Multiphysics

Yashashri Patil[1], D. Krishnan[1], and S.S. Khirwadkar[1]
[1] Institute for plasma research, India
*Corresponding author: Bhat Gandhinagar-382428, Gujarat, ypatil@ipr.res.in

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. 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.
3D and 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. Stresses developed during transient heat loads at various parts of mock-up have been calculated using structural mechanics.
Keywords: Plasma facing component, thermo mechanical analysis, Divertor

1. Introduction

Plasma facing components of a divertor are fabricated in duplex structure. Carbon (Graphite/CFC) and Tungsten (W) are the plasma facing materials (PFMs) joined to the actively cooled CuCrZr alloy heat sink. Vacuum brazing is most common joining technique used for development of such duplex geometry (as shown in figure 1) [1]. Divertor is act as an exhaust for incoming heat flux from the fusion plasma. According to its function surface heat flux incident on PFMs are actively transfer via a heat sink to coolant. Heat Transfer performance of developed mock-up is qualified using High Heat Flux (HHF) test. During HHF test the developed mock-up is exposed to steady state and transient high heat flux ~ 10 MW/m^2 using High energy electron beam. Manufacturing process like high temperature vacuum brazing develops residual stresses in developed mock-up and in addition to that thermal stresses are developed during transient heat loads (HHF test). Hence Thermo-mechanical behavior of the divertor test mock-ups under transient heat loads is a crucial step in development of divertor Plasma facing components.

In this work, 3D thermal profile developed for graphite mock-up under 7.22 MW/m^2 transient heat loads is simulated using heat transfer module. Residual stresses developed during high temperature vacuum brazing and thermal stresses developed during HHF test are simulated using structural mechanics module. Simulation of whole mock-up geometry consumes lot of computational time and memory. Hence semi-model of mock-up with six graphite tiles is developed.

2. Numerical Model

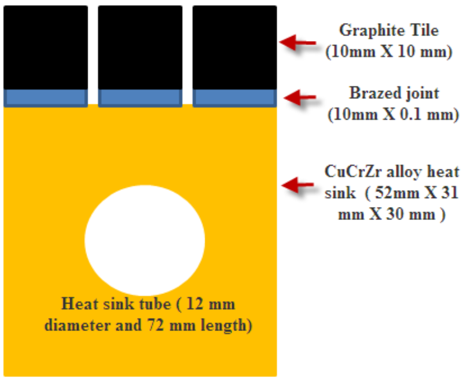

Figure1. Schematic diagram and dimension of Graphite test mock-up

2.1 Geometry

Schematic of Graphite mock-up with dimensions used for the study is as shown in figure1. It consist 15 square graphite tiles brazed with rectangular CuCrZr alloy block. Block is having central water channel or tube. Whole assembly was drowned using Comsol multiphysics geometry toolbox. Whole geometry is divided in 31 sub-domains. Fifteen sub-domains are assumed as graphite tiles and Thermo-physics properties of graphite are assigned to it. Fifteen sub-domains are assumed as brazed joints. One rectangle with tube is assumed as CuCrZr alloy heatsink. All the material properties used for the model are summarized in Appendix.

2.2 Governing Equations

Temperature profile on Graphite mock-up can be obtained by solving conduction equation in Cartesian coordinates.

$$\rho Cp \frac{\partial T}{\partial t} = \nabla.(k\nabla T) + Q - - - (1)$$

Where, ρ = density of material, Cp=Specific heat capacity, k = Thermal conductivity of material, Q = heat flux, T(x, y, z, t) = temperature In this problem 'Q' is composed of incident heat flux(q_0) from the electron beam and convective heat flux due to cooling of mock-up through heat sink[2].

2.3 Boundary Condition Formulations

The boundaries conditions that apply to the model are heat flux at surface of Graphite tile and thermal insulation at other surfaces of mock-up are given by equations 2& 3

$$-\vec{n}.(-k\nabla T_1) = q_0 + h(T_{\inf} - T_1) - - - - (2)$$
$$-\vec{n}.(-k\nabla T_1) = 0 - - - - (3)$$

Where *n* is the normal vector of the boundary, q_0 The inward flux heat, *h* the heat transfer coefficient, T_{inf} the external (ambient) temperature, and T_1 the initial temperature. Radiation losses by mock-up are not considered in this problem. Hence the Heat flux boundary condition q_0 is considered as extracted heat flux by mock-up 7.22 MW/m^2 which corresponds to incident heat flux 10 MW/m^2 for *h*=0. Note also that the interface of the mock-up and the cooling water chamber is an internal boundary common to all graphite tiles domains for which $q_0 = 0$.

Internal boundaries of water channel are assigned as forced cooling by water having temperature 100°C for which boundary conditions are h = 50,000 [W/m²K] and $q_0 = 0$.

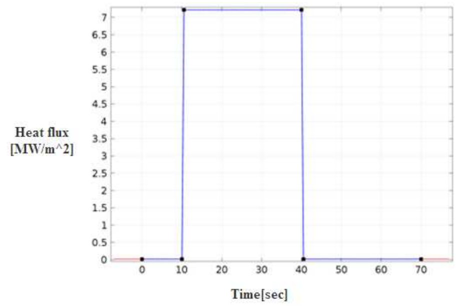

Figure2. Incident heat flux pulse

Figure2 shows Input incident heat flux is applied at top surfaces of graphite tiles. For stress analysis the Brazing cooling curve has assigned to all boundaries of mock-up. Fixed Constrains are apply graphite tiles and stresses developed during HHF test are calculated using temperature profile.

2.4 Mesh conditions

Automated Free triangular mesh is used for the solution. Complete mesh consists of 5573 elements.

2.5 Solver conditions

The linear solver UMFPACK is used for the problem. Time is set for 0 to 80 sec with 0.1 sec time step.

4. Results and discussion

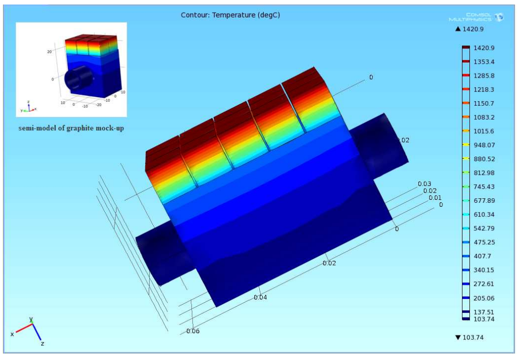

Figure3. 3D contour plot of temperature profile at 31 sec

Figure3 shows temperature profile of Graphite mock-up during transient heat load with T_{max}=1420°C and T_{min}100°C.

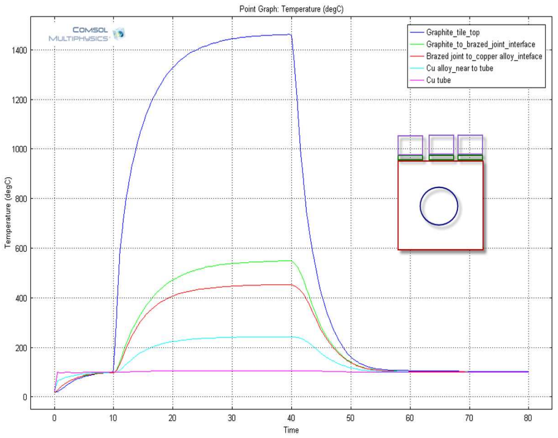

Figure4. Temperature VS time plot test mock-up during HHF test

Figure4 shows the temperature observed at the various domains which having different thermo-mechanical properties. Figure4 gives information about steady state temperature developed on graphite mock-up. Maximum residual stresses observed on mock-up after brazing cycle are 946 MPa. Maximum stresses developed during transient HHF loads are approx. 1208 MPa.

5. Conclusions

The paper deals with the numerical modeling of Graphite mock-up during transient heat loads, which used for validation of experimental results. Maximum surface temperature observed during simulation is 1420ºC.

Singular residual stresses developed after brazing may be the reason behind the tile failure. Thermal stresses developed during HHF tests are calculated using thermal profile observed on mock-up. Detailed Residual and thermal stress analysis will be carried out in future.

6. References

1. R. Giniatulin, High heat flux tests of mock ups for ITER divertor application, *Fusion Engineering and Design* **39-40**, 385-391(1998)

2. Luděk, Fe model of thermo-mechanical interaction in rubber block under dynamic stress *Proceedings of the COMSOL Users Conference 2006 Prague, (2006)*

3. J.H. You, Analytical method for thermal stress analysis of plasma facing materials, *Journal of Nuclear Materials*, **299**, 9-19 (2001)

4. User manuals of Comsol multiphysics

7. Acknowledgements

The author would like to thanks Director of IPR Author would like to acknowledge Amardas Ali for valuable guidance.

8. Appendix

Material properties of Graphite:
Density of graphite =1.820 g/ cm^3
Poisson's Ratio=0.18

Temperature [ºC]	Thermal conductivity [W/m K]	Specific Heat (J/kg K)
20	155	719.78
300	118	1379.12
500	92	1598.9
800	69	1802.19
1200	53	2021.97
1600	47	2098.9
2000	40	2159.34

Temperature [ºC]	Thermal expansion coefficient [/ºC]	Young modules [Pa]
20	4.70E-06	9.30E+09
300	5.10E-06	9.40E+09
500	5.50E-06	9.50E+09
800	5.60E-06	1.03E+10
1200	6.00E-06	1.08E+10
1600	6.50E-06	1.14E+10
2000	6.90E-06	1.22E+10

Material properties of CuCrZr alloy:
Density of CuCrZr =8.96 g/ cm^3
Poisson's Ratio=0.33

Temperature [ºC]	Thermal conductivity [W/m K]	Specific Heat [J/K. kg]
20	384	3.78E+02
200	372	4.05E+02
400	359	4.25E+02
600	355	4.43E+02
800	350	4.58E+02

Temperature [ºC]	Young modules [Pa]	Thermal expansion coefficient [/ºC]
20	8.24E+10	0.0000154
200	7.36E+10	0.0000166
400	6.87E+10	0.0000183
600	6.18E+10	0.00002
800	5.89E+10	0.0000216