COMSOL CONFERENCE INDIA 2012

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

Presented by

Yashashri Patil

D. Krishanan, S. S. Khirwadkar

Institute For Plasma Research , Bhat

Gandhinagar-Gujarat-382428

ypatil@ipr.res.in

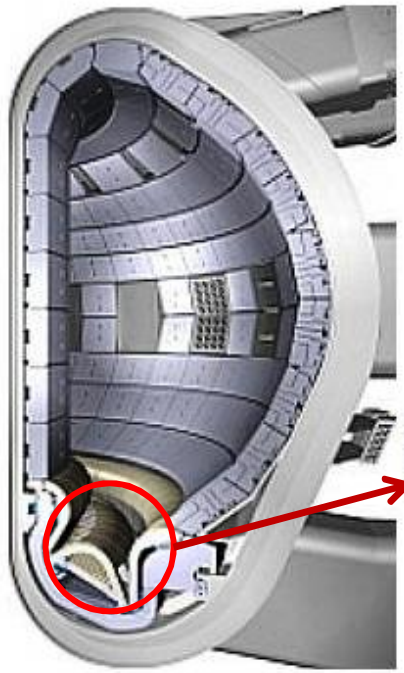
3 November 2012

Excerpt from the Proceedings of the 2012 COMSOL Conference in Bangalore

Outline

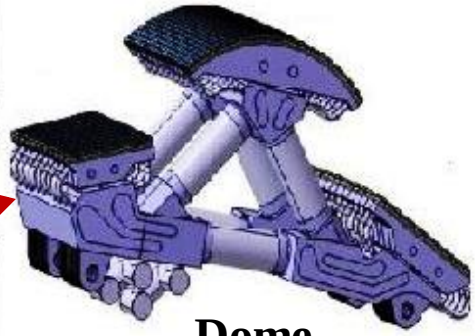
- ✓ Introduction and motivation
- ✓ Mathematical model
- ✓ Thermal properties of a brazed joint using Comsol
- ✓ 3D Comsol model for HHF test & boundary conditions
- ✓ Results
- ✓ Conclusions

Introduction and motivation



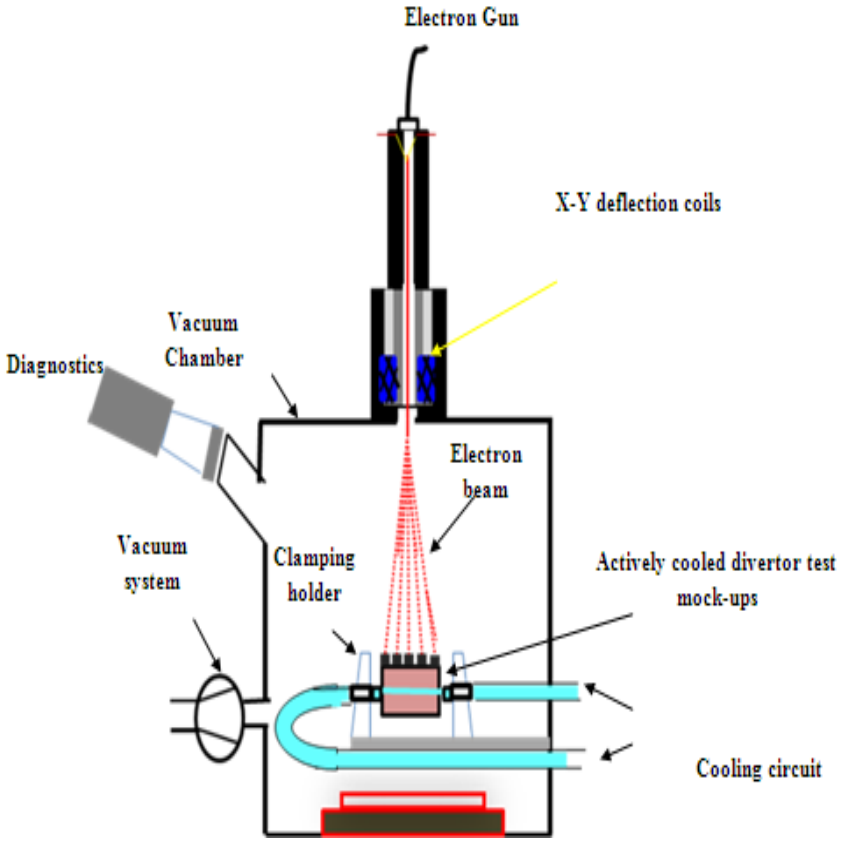
Divertor

Graphite mock-up



Dome

Vertical cross section of ITER like fusion reactor & Dome of divertor (www.iter.org)

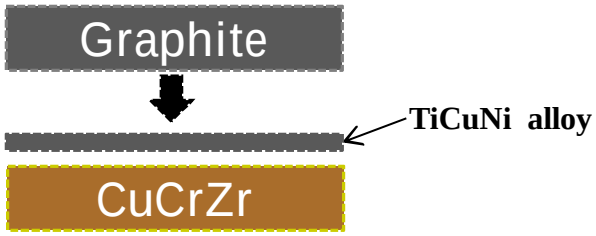


Schematic diagram of High heat flux(HHF) test set-up

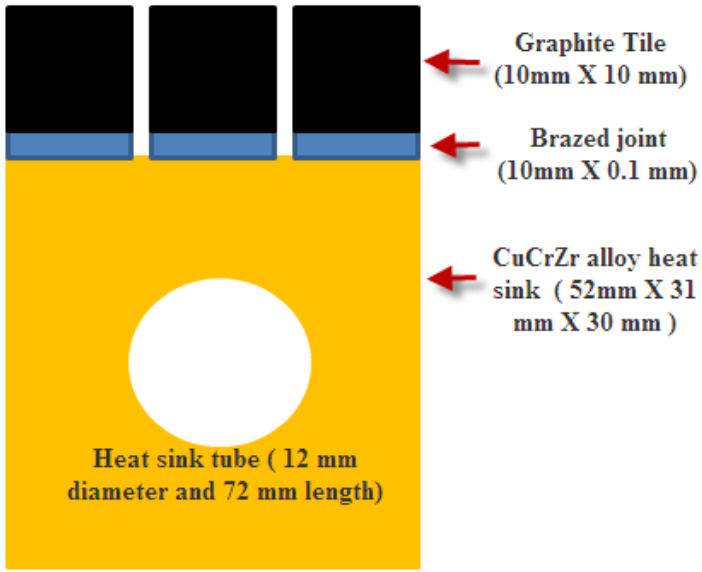
Plasma facing components namely Inner vertical target, Outer vertical target, Dome of divertor is exposed to a thermal load of 5MW/m^2 to 20 MW/m^2.

Graphite macro-brush type mock-up was exposed to 10 MW/m^2 using High energy electron beam HHF facility.

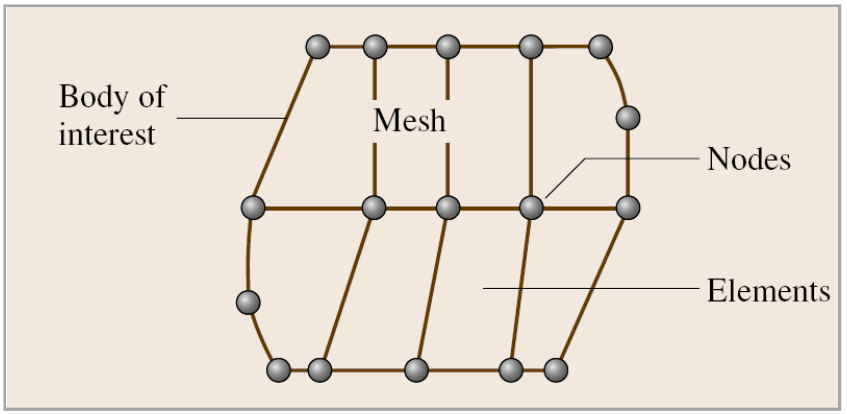
Mathematical model



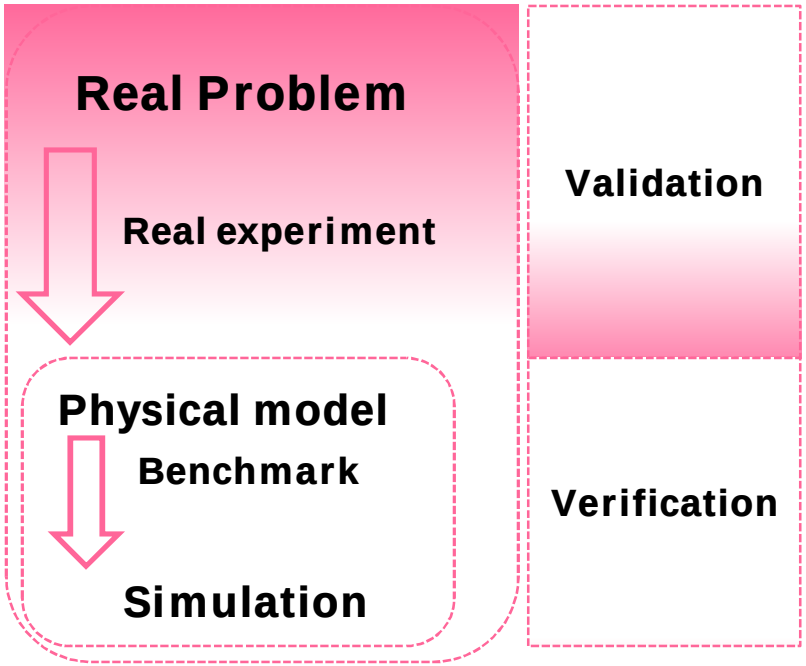
Graphite Tiles were brazed with heat sink using TiCuNi-70 @980°C



Schematic diagram of Graphite mock-up



Typical Finite element mesh



Mathematical model

Governing Equation

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

Boundary Condition

Heat flux

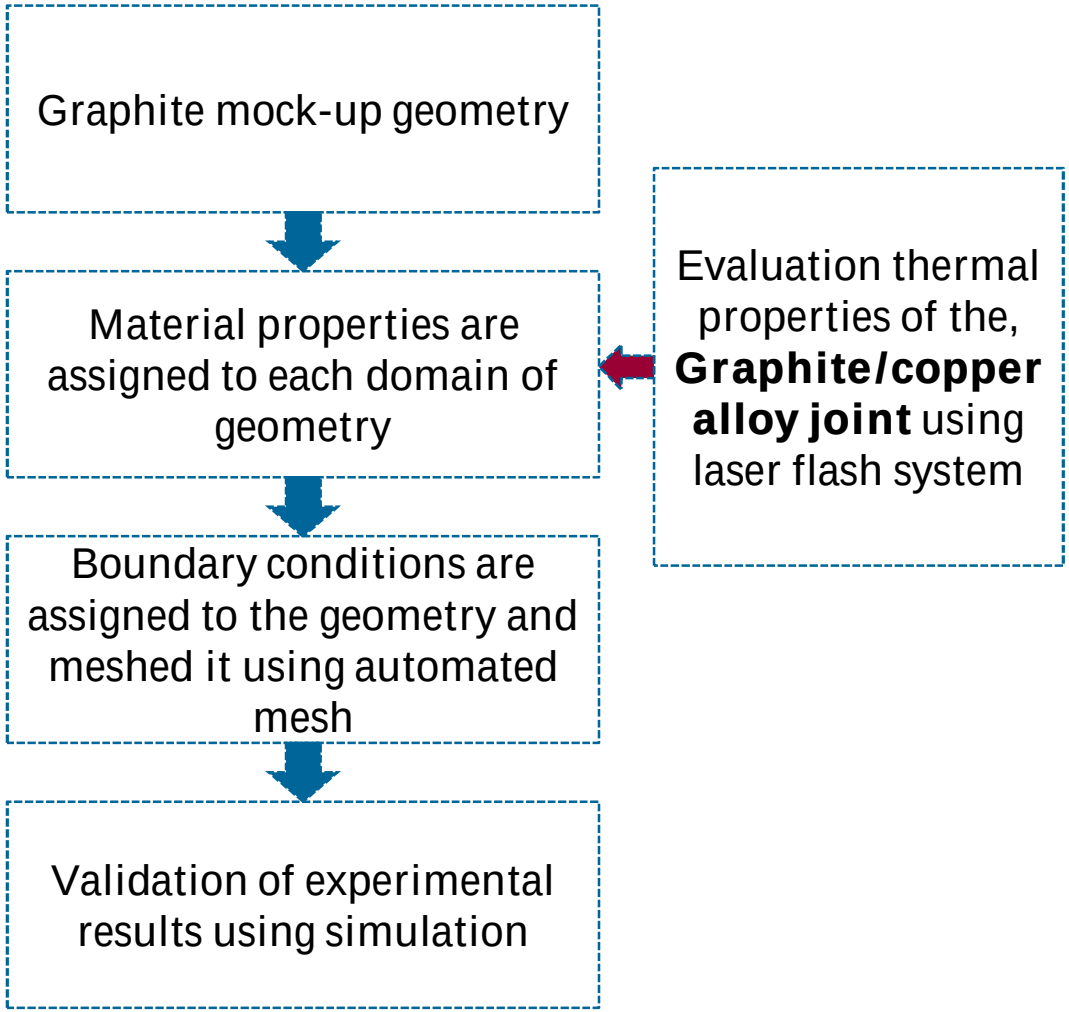
$$-\vec{n}.(-k\nabla T_1) = q + h(T_{inf} - T_1)$$

Thermal insulation

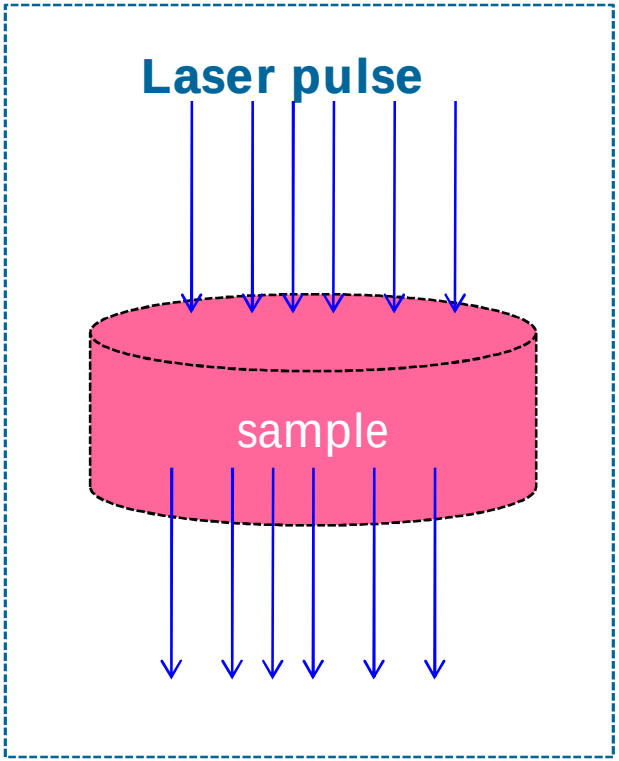
$$-\vec{n}.(-k\nabla T_1) = 0$$

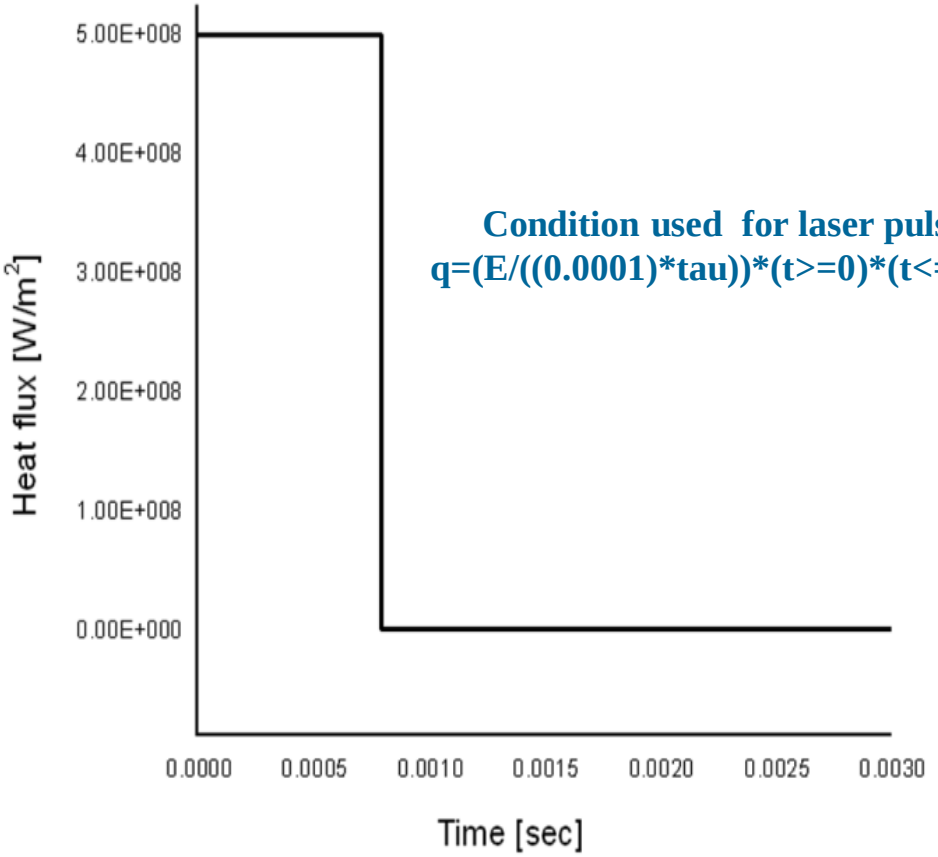
Where,
ρ= density of material,
Cp=Specific heat capacity,
k = Thermal conductivity,
Q= heat source
q = heat flux, h =heat transfer coefficient
T(x, y, z, t) = temperature, T_{inf} =external temperature , T1=initial temperature

Model flow chart



Thermal properties of a brazed joint using Comsol





Condition used for laser pulse
q=(E/((0.0001)*tau))*(t>=0)*(t<=tau)

Laser pulse used for Comsol model

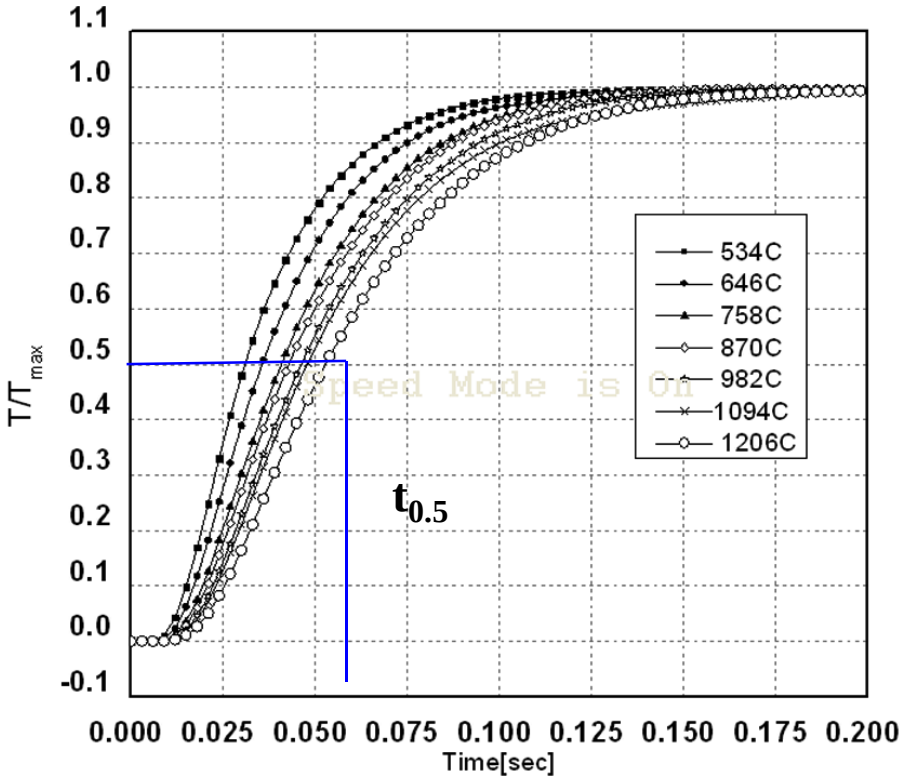
$$\alpha = \frac{0.1388\, L^2}{t_{0.5}}$$

Where, α: Thermal diffusivity,
L:Thickness of sample, $t_{0.5}$: t half

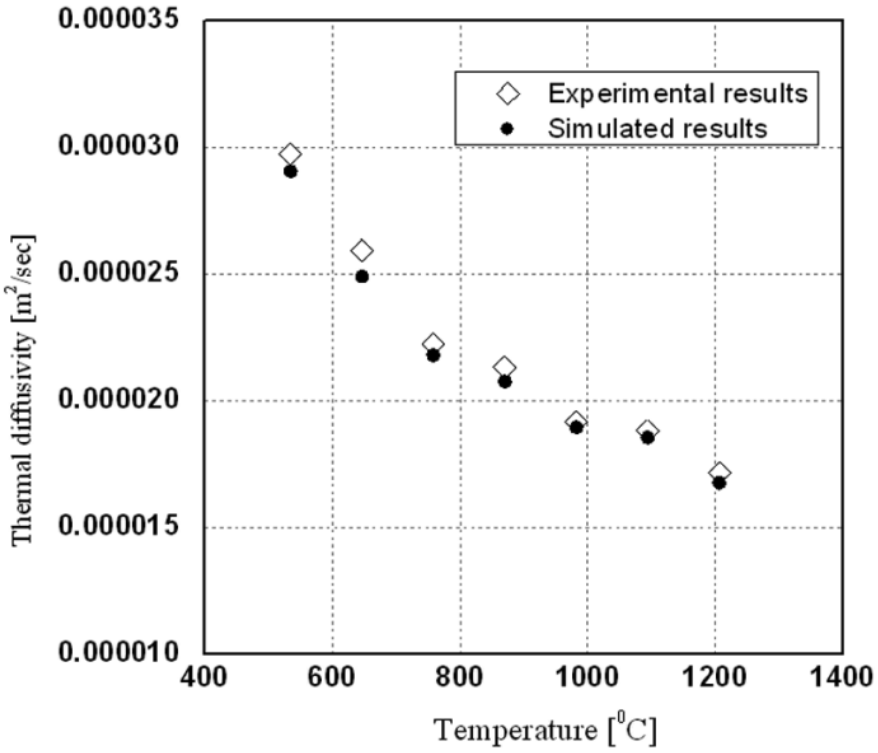
Pulse energy (E) = 40J ,
Pulse width(tau)=0.8milisec,
P_{in} = 5 x10 ^{8} W/m^2

W.J. Parker, et al.; Journal of Applied Physics, Vol.32 (1961) p1679

Thermal properties of a brazed joint using Comsol (Benchmark)



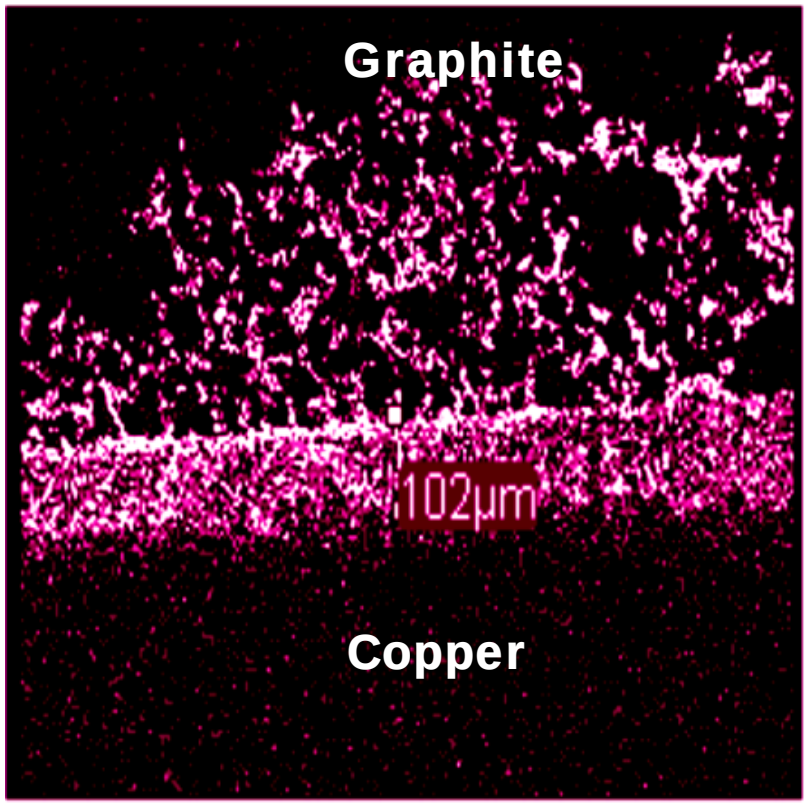
T/T_{max} Vs time plot generated of FP479 Graphite using Comsol Multiphysics



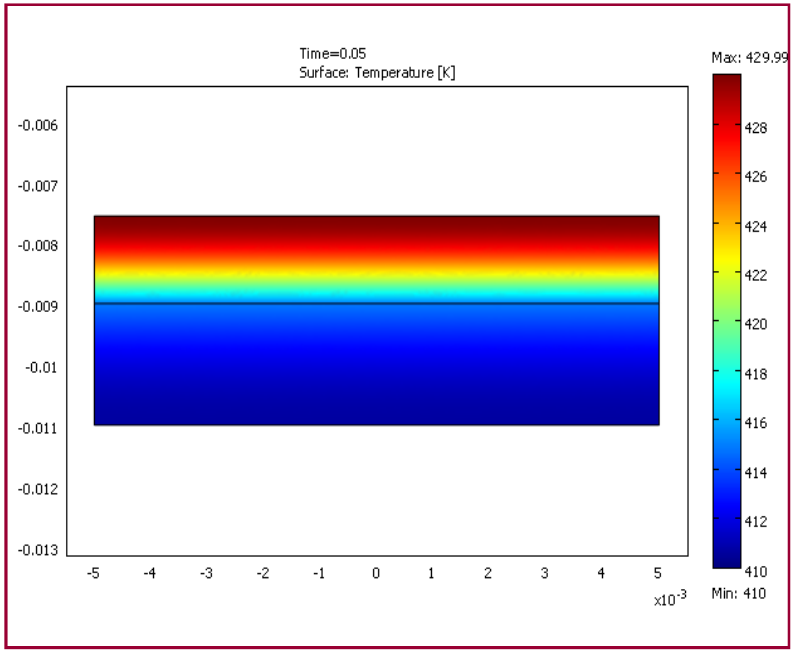
Comparison experimental results with simulation results

Simulation results are nearly match with experimental results.

Thermal properties of a brazed joint using Comsol



SEM of Graphite/TiCuNi/Copper joint



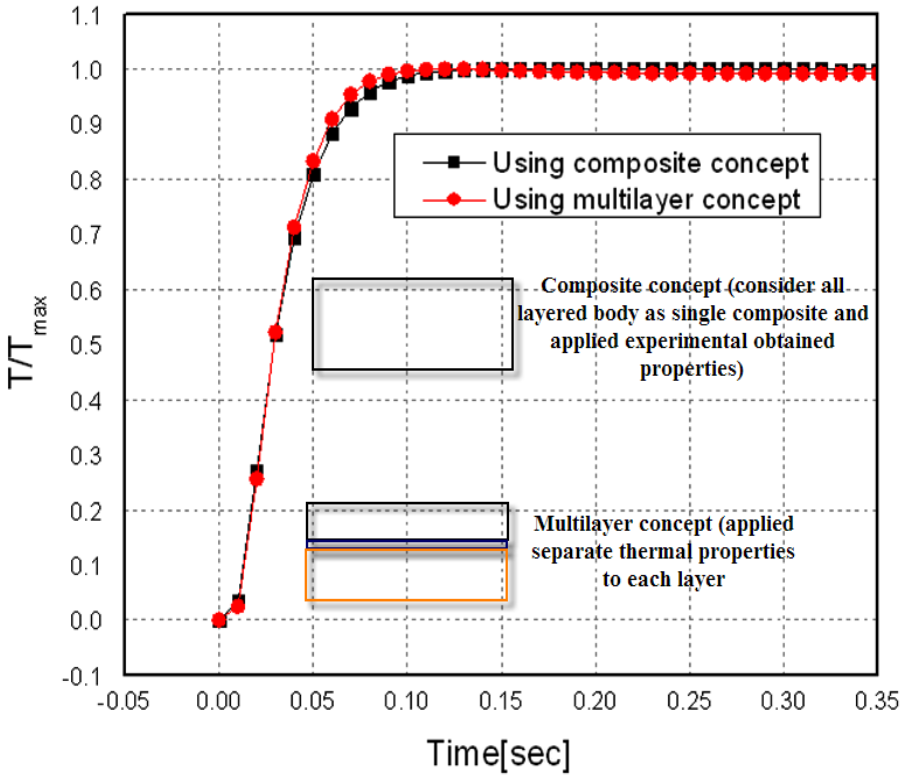
2D model of Graphite/copper joint

$$Q = \frac{\Delta T}{\frac{d_1}{k_1} + \frac{d_2}{k_2} + R_{joint}}$$

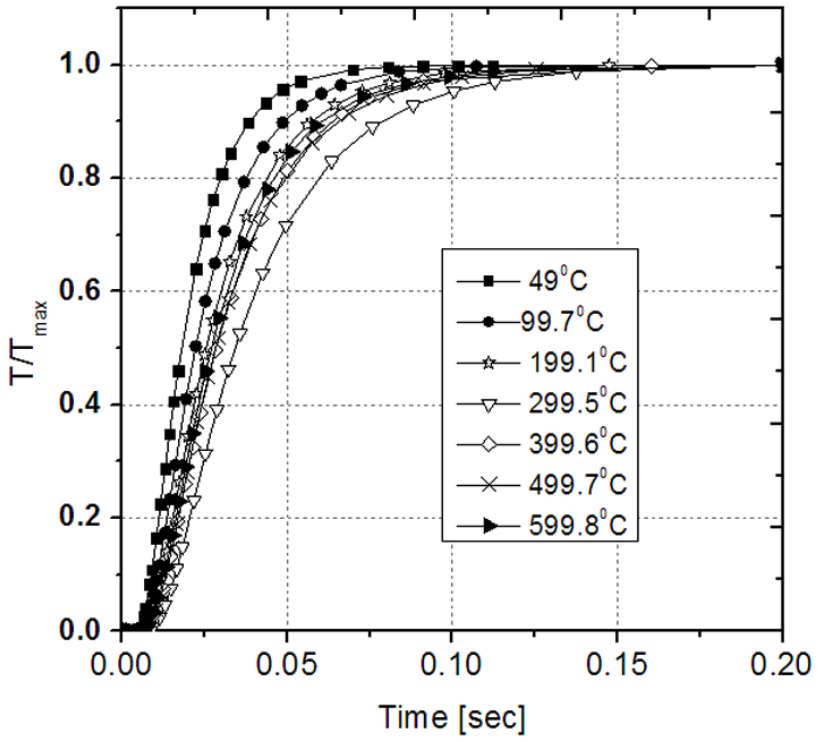
2D model of Graphite/ Copper joint was developed in Comsol Multiphysics and using iterative method thermal properties of the brazed joint was evaluated for temperature range 50˚C to 600˚C.

ypatil@ipr.res.in

Thermal properties of a brazed joint using Comsol



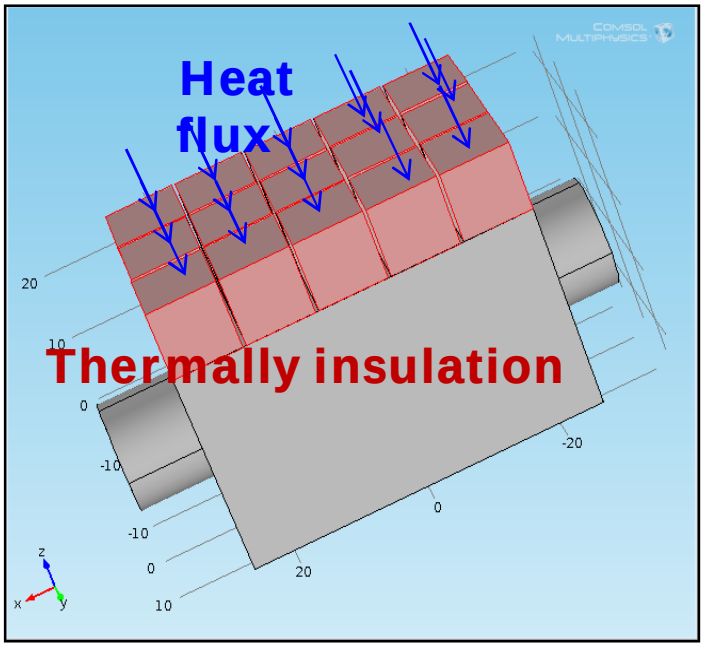
Raw plot generation using experimental thermal properties

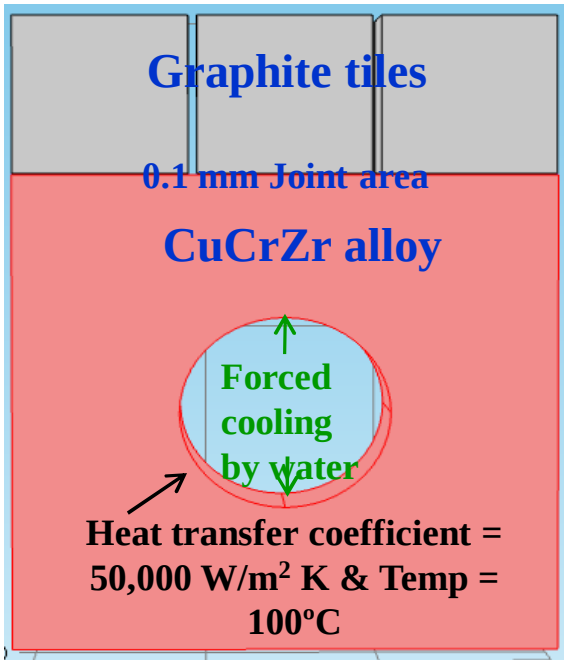


T/T_{max} Vs time plot generated using Comsol Multiphysics

Thermal conductivity of joint interlayer is in the range 16 W/m K(50°C) to 5.4 W/m K(600°C) and further used for HHF model.

Comsol model & boundary conditions





Input Heat flux is an extracted heat flux by mock-up 7.2 MW/m^2

Heat Transfer Coefficient h = Nu. k / d

Where, **Nu = 0.023*Re$^{0.8}$*Pr$^{0.4}$; Pr = μCp/k; Re = ρ Vd/μ; m = ρ AV**

Graphite

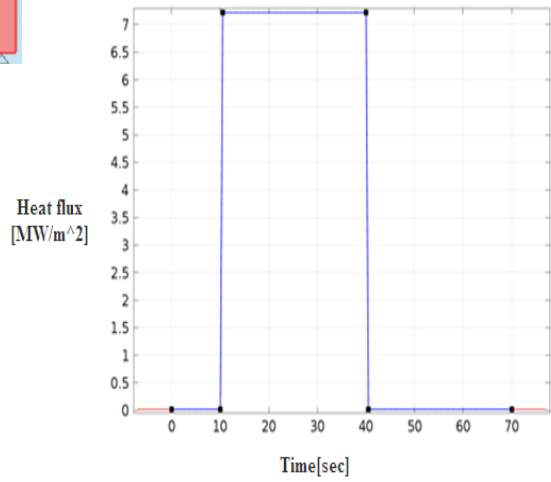
Temperature [°C]	Thermal conductivity [W/m K]	Specific Heat (J/kg K)
20	155	719.78
300	118	1379.12
500	92	1598.9
800	69	1802.19
1200	53	2021.97
1600	47	2098.9
2000	40	2159.34

Density = 1.820 gm/cc

CuCrZr alloy

Temperature [°C]	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

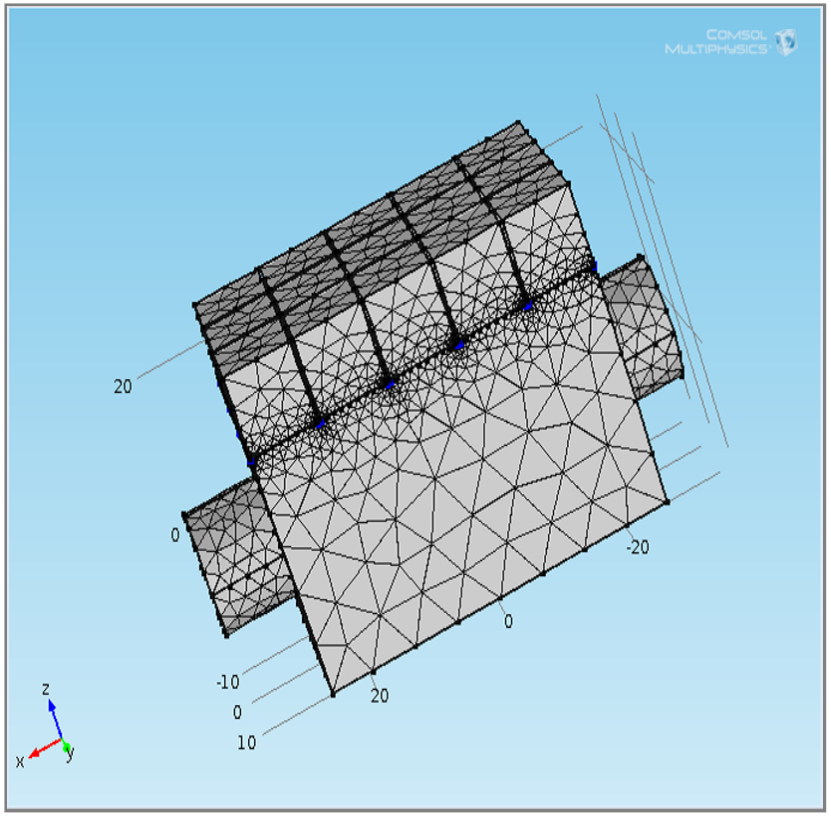
Density = 8.96 gm/cc



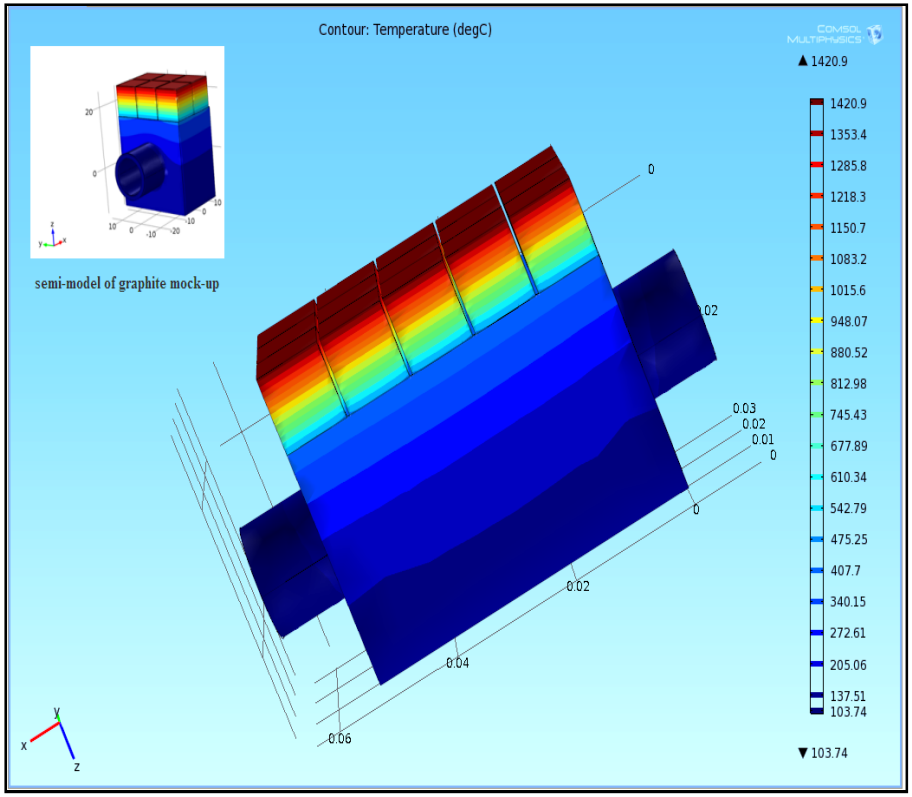
Incident heat flux

ypatil@ipr.res.in

Results



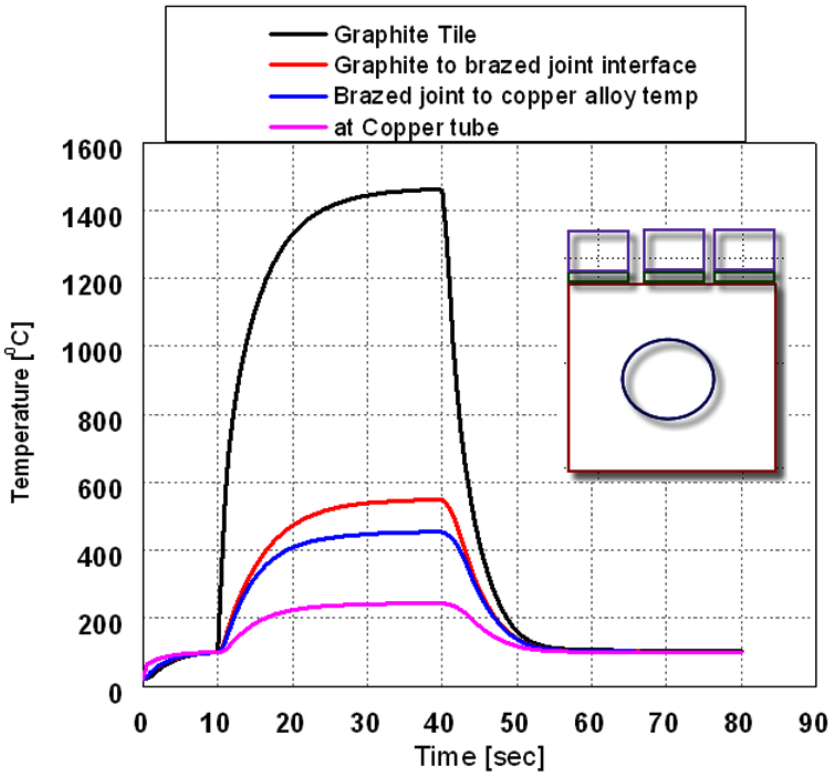
Complete mesh consists of 40799 elements

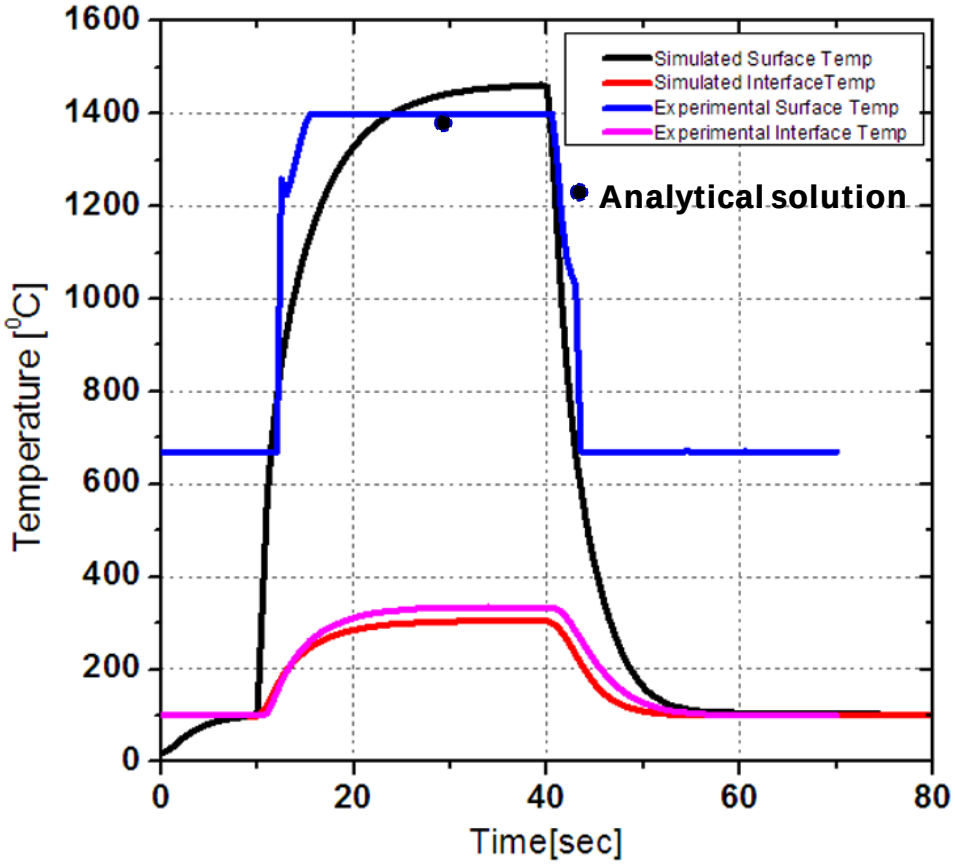


3D contour plot of temperature profile at 31 sec

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

Results





Analytical surface temperature of a mock-up is calculated using 1D steady state conduction equation q= k dT/dx.

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.

Acknowledgements

The author would like to thank Director of IPR also author would like to acknowledge Amardas Ali for valuable guidance. The authors would also like to thank all the members of Divertor & First wall Technology Development Division.

THANK YOU