EL044A - LAB02

Objective

The objective of this lab will be to model heat sink, thermal stress, convective cooling and forced convection.

TASK – 1 – Modelling of thermal resistance $R_{\theta jc}$

- Open the model given in the moodle page – "Chip_model" with one IC and copper tracks on the FR4 board. Setup the boundary conditions for a heat transfer model as given in lab01.
- Add an additional thermal contact boundary condition on between the IC chip and the component i.e. R_{jc}, with the value of 4.2 degC/W as shown in figure-1

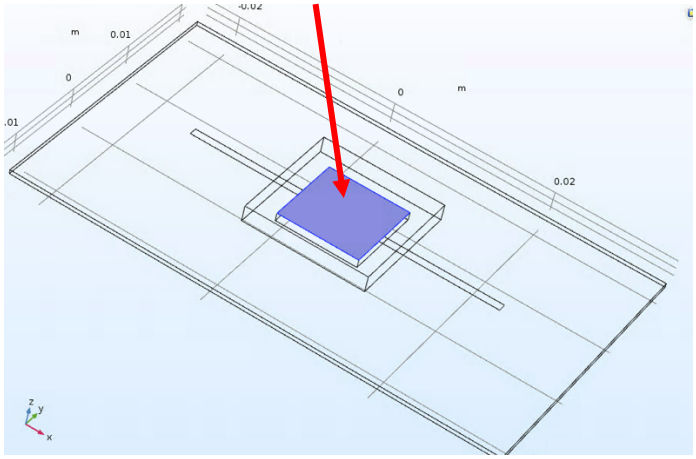

Figure-1: Thermal contact R_{jc}

*Hint: needed boundary conditions are: Heat source, heat flux (both top and bottom plate).

- Run the simulation with 0.5 W of heat source and 1 W of heat source.

"Use the cut line surface normal" result feature to observe the maximum junction temperature and save for the report. And explain your findings and method to the instructor.

TASK – 2 – Modelling of heat sink and resistances $R_{\theta ch}$ and $R_{\theta ha/}$ $R_{\theta sa}$

- From the parameters list enable the heat sink by setting the parameter to 1 from 0.
- After the heat sink is available select the material as aluminum for the heat sink
- Select a thermal contact between the surface of the heat sink and the IC and give a value of 5.5 degC/W as shown in figure-2.
- For the heat sink select a heat flux boundary condition and select convective heat flux and then user defined heat transfer coefficient of 5 $W/m^2.K$.

- Observe the temperatures T_j, T_c and T_h – explained in lectures 1 and 4.

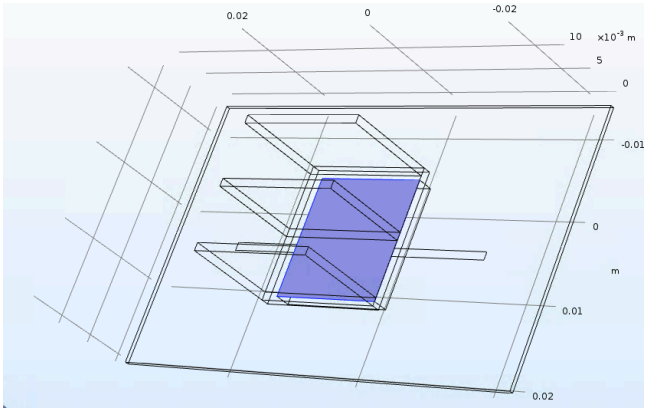

Figure -2: Thermal Contact R_{CH}

- Change the following parameters given in the parameter list to get the temperature of the junction below 55 degC.
  - Fin width – "fw"
  - Fin height – "fh"
  - Number of fins – "nf"

After the tests, there should be more options than one to get the required junction temperature. Analyze and save them for the report and explain to the instructor what you observed.

TASK – 3 – Thermal stress analysis

- Download and open the model available on the moodle page "Thermal stress"
- Run a parametric study on the variable "Psource" from 0.2 to 0.6 W with a step of 0.05 W and document the von mises stress on the board.
- Understand the model boundary conditions using COMSOL help (F1) and document your description of the model in the report and your understanding of the model setup.

TASK – 4 – Forced convection with a heat sink model

- Download and open the model "Forced convection" from the moodle page.
- Run parametrized study on parameter "air_speed" that is used in the "inlet" boundary condition and document the temperature drop on the heat sink at different input speeds from 4 to 8 m/s with a step of 1 m/s.
- Change the heatsink parameters as given in task 2 and analyze the change in results. Select these parameters based on your own developed understanding from task 2 to get the temperature on the fin top surfaces to below 60^{o}C.
- Document the results for all the steps for the report and your conclusions about the methods and findings.

Report

A short report of the lab results should be submitted including method details and obtained results, with clear conclusions about each task. Along with the report the original COMSOL work files should be submitted before **the deadline** on the course Moodle page.

**Assignment # 02 hint: Open up the geometry in the model and see how the heat sink is designed if you want to test your findings in COMSOL for pin and triangular type heat sinks.*