Objective

The objective of this lab will be model multiple components on an circuit board and apply electrical excitation and analyze the thermal distribution. It will introduce the Multiphysics coupling and applying boundary conditions.

TASK – 1

Create a single component thermal simulation with parametrized heat source ranging from 0.2 W to 1 W.

*Note: For steps 1-4 tutorial-1 is also available.

STEP-1

- After opening the COMSOL software, select **Model Wizard**
- Then **3D → Heat transfer → Heat transfer in Solids (ht)**
- After that on the bottom right **click the Study button**
- From the preset studies select the **Stationary study** and **click Done** on the bottom of the window. Now your COMSOL window should be as shown in figure-1
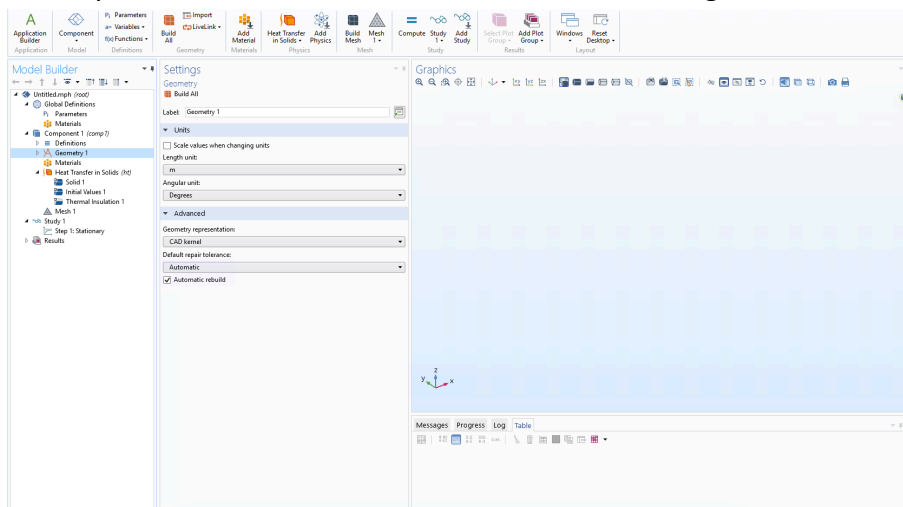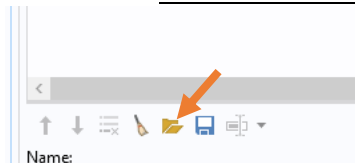

Figure.1 – COMSOL Startup window

STEP-2

Under Global definations, **select the parameters**
In the parameters settings window click the **load from file icon**

From the downloaded lab folder select the "**Para_list**" file and click open

STEP-3

- In the settings window **change the length unit from m to mm from the drop down list**
- Right click the Geometry 1 under the model tree and **select insert sequence**
- Select the file **FR4-150x120x4.mph** from the saved folder and select built.
- Right click the Geometry 1 under the model tree and **select insert sequence**
- Select the file **IC-10x10x1.8.mph** from the saved folder and select built.
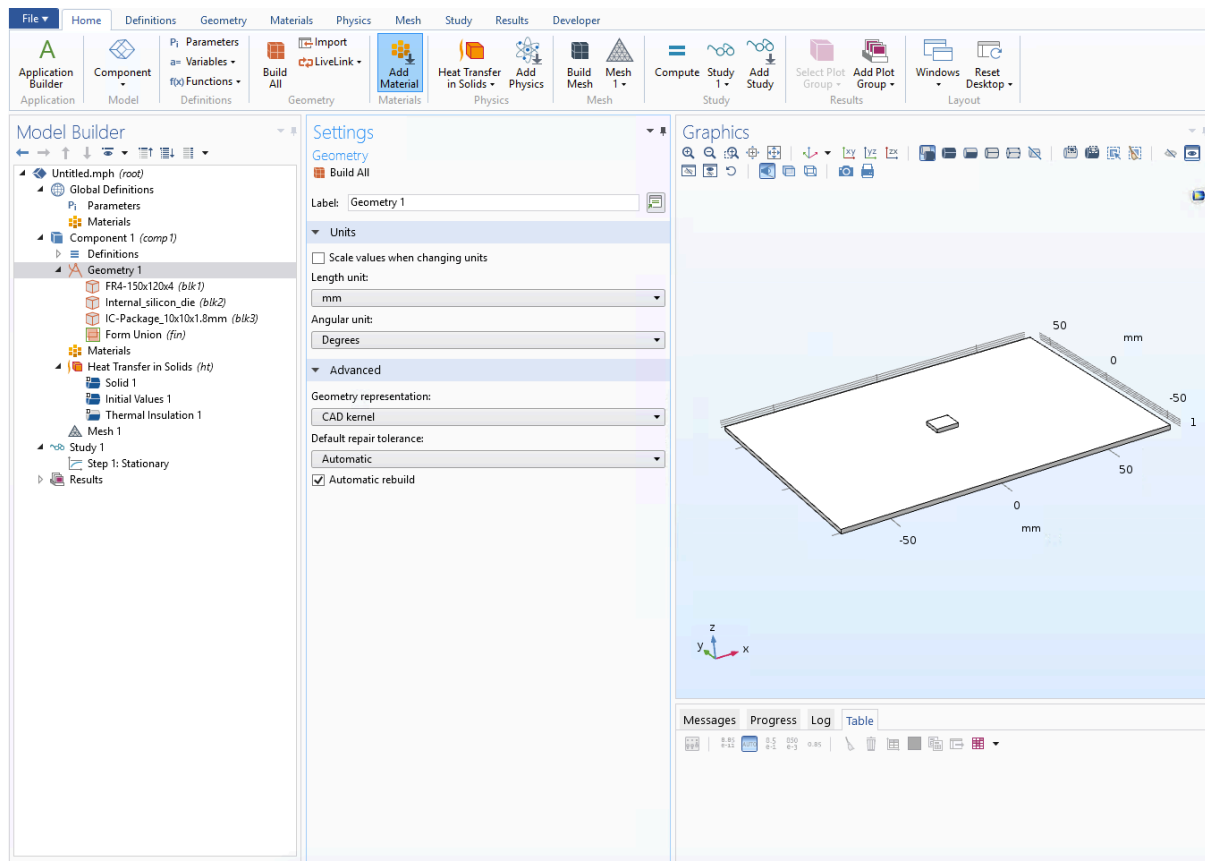
Now your window should be as shown in figure.2

Figure.2 – After the built of the geometry

STEP-4

- Right click the materials in the model tree and select **add material from library**
- Double click the following materials from the **built-in** tab to be added to your component
  - ➢ FR4 (Circuit board)
  - ➢ Silica glass
  - ➢ Acrylic plastic
- Assign the materials to the respective geometric domains using various view options.

STEP-5

Then set the boundary conditions for the heat transfer module as shown in the tutorial-2.

STEP-6

Under the mesh settings select Physics controlled mesh and from the drop down select finer.

STEP-7

In the study node select compute.

STEP-8

Analyze the results for thermal distribution on the board and a cutline through the IC.

STEP-9

- Add a new stationary study to the model tree, right click it and add parametric study node.
- In the parametric study settings click the + button and select the Pt_IC from the list and set the range to be 0.2 to 1 W with the step of 0.1 W.

STEP-10

Using the steps explained in **step-3** import 2 new IC components; IC2 and IC3 from the provided LAB folder.

Change the placement of the new ICs using the parameters IC2_x and IC2_y and respectively for the third IC IC3_x and IC3_y, such that they are parallel to each other in y-axis, with a separation of 15mm.

STEP – 11

Using the same condition of heat source under the Heat transfers in solid apply 0.5, 0.7W and 0.5W respectively to the three IC components.

STEP – 12

For the reporting of results from step-12 create a table with Input power Vs max temperature.

Analyze the results and save the file and explain to the instructor.

TASK – 2

Apply the thin layer feature as explained in the lecture-3 to the three IC components base attachment to the FR4 board. See also introduction to heat transfer guide. Page-17 and 42.
Use the Gap conductance as 3.1 degC/W
Note that the unit in the settings window is W/K.m, this means that your input gap conductance will be:

$$1/\text{area of the contact} * 3.1[\text{degC/W}]$$

Analyze the results and save as a new file and explain to the instructor.

TASK − 3

Apply electrical parameters to the added components.

STEP − 1

After opening COMSOL; **3D** → **Heat transfer** → **Electromagnetic heating** →**Joule heating AC/DC** →**Electric Currents, Shell (ecs)**

STEP − 2

Follow the step 3 in task 1 to import the geometry and tutorial-1 to select the materials

STEP − 3

Make the copper tracks according to tutorial-3

STEP − 4

Add a new boundary material copper to the materials.

STEP − 5

Set the boundary conditions and physics interface according to tutorial-4: Note that which domain requires which physics interface.

STEP − 6

Run the simulation with varying power input at 5V and analyze the results and explain to the instructor.

STEP − 7

Apply different input powers and analyze the results.

STEP − 8

Set the thermal contact settings and visualize the changes.

STEP − 9

Run a parametrized study for the track depth between 180 um to 300 um with a step of 10 um.

STEP − 10

Run a parametrized study for the contact resistance between 2 degC/W to 4 degC/W with a step of 0.25 degC/W.

STEP − 11

For the report create a table with Track_depth Vs Max temperature and Contact resistance Vs Max temperature.

Analyze the results and save as a new file and explain to the instructor.

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.