THE FINITE ELEMENT METHOD

HEPATIC TUMOR ABLATION

LABORATORY EXERCISES

BIOMEDICAL MODELLING AND SIMULATION

TBME08

Department of Biomedical Engineering
Linköping University

Tutor:	Neda Haj-Hosseini
Place:	Zoom (See the link in Lisam)
Software requirement:	Remote access to US- hus 462 computer lab (IMT8)

Introduction

The exercises in this compendium should be performed using COMSOL Multiphysics version 5.6 (latest version) together with the model provided. You will need to start with going through the model and locating different parts of the model. The results of the tasks should be submitted as a written report and results should be motivated by the logics and why you consider them to be correct. Include an introduction to the mathematical model at

Save the changes to the program once you are certain of the performance. This enables you to re-use them without having to redo the changes.

Reports are written in a group of two persons (one report per group) and should be uploaded in pdf format in LISAM latest **10 days** after the lab session.

Preparatory lab quiz should be done on Lisam after the lecture and before the lab.

Task 1. The model

Navigate in the model, find and write down the Multiphysics modules used in the model.

a. What are the governing equations and field variables for each module?

   1 In the model **Builder window**, locate the **components**.
   2 Find the physics modules and the **equations** under the corresponding **Setting** window.
   3 Locate the **Dependant variable**

b. The model domains: Find the implemented domains in the model geometry, the initial conditions and boundary conditions of the domains for the modules implemented. Describe these in the report and motivate the reason for the choice.

   To find the domains:
   1 In the **model builder** window, locate **Definitions**, **Selections**.
   2 Locate the implemented domains in the model geometry by clicking on each domain.

   To find the initial and boundary conditions:
   1 In the **components,** locate the physical modules and relate the initial and boundary conditions to the domain.

Task 2. Meshing

Before running the simulation, you will need to choose a suitable mesh size. Try at least two optional mesh sizes. Select the one that suits the simulation best by comparing the mesh quality versus computational time and the application.

To change mesh size:
1 In the **model builder** window, locate components, double click on **mesh 1**, select **size.**
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 Choose a predefined or custom size.
4 Click **Build All**.

To obtain mesh quality measures:
1 Right click on **Mesh**, derive **Statistics**.
2 Use the **minimum** and **average mesh quality** and the histogram to evaluate the mesh.

Task 3. Domain probe in the blood vessel (5)
The tumor should be heated and damaged without damaging the blood vessels. Place a probe close to the blood vessel to monitor its temperature.

1 In the **Definitions** toolbar, right click on **Domain Point Probe 4, duplicate.**
2 In the **Settings** window for **Domain Point Probe 5** (the new probe), locate the **Point Selection** section.
3 Change **Label** to Domain Point Probe 5-vessel.
4 In **Coordinates**, set **x, y, z** so the probe is located in the liver tissue adjacent to the vessel.

Task 4. Simulation
Perform a time-dependant simulation. Set the time interval and time steps for the output results.

1 In the **Model Builder** window, locate **Parameters**, change *a_time* to 30 mins.
2 In the **Model Builder** window, under **Study** select only **Generate default plots**
3 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.
4 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
5 In the **Output times** text field, type range (0,a_time/10,a_time).

Task 5. Result visualization- Damaged tissue volume
The tissue heated above 45-50 deg C is considered as ablated in this medical procedure. Find the volume of the damaged tissue by visualizing the volume of the liver tissue heated above this temperature.

a. Plot the volume of the heated tissue in 3D.

1 In the **Model Builder** window, locate **isothermal contours** and visualize the volume with temperature above 50 degC.
2 Save the graph and enclose to the report.

b. Plot the fraction of the damaged tissue at the three probe positions in the liver tissue as a function of time.

1 In the **Model Builder** window, locate **Results**, **Damaged tissue, 1D**.
2 Right click, in the **setting** window, locate **Data section, columns,** select the **Data** (Probes 1-3)**, click plot.**
3 Right click, in the **setting** window, locate **Data section, columns,** select the **Data** (Probes 4-5)**, click plot.**

c. Find the actual volume of heated tissue.

1 In the **Model Builder** window, right click on the **Derived values**.
2 Add **volume integration**. Locate the **expression** section.
3 Locate **Expressions**. Add **if(T> 50+273, 1,0)**.
4 Set the unit of the volume to your desired scale. e.g. mm^3
5 Click on **Evaluate**. Check the results in the table generated.

d. Suggest changes in the model parameters without changing the geometry to compromise between maximum tumor tissue ablation (T> 50, fraction of damage above 50%) and avoiding damage to the vessel (T< 40). Motivate your choice in the report. Assume that the tumor has a radius of 15mm with its center being in the middle of the electrode trocar.

Hint: Use the domain probes 4 and 5 for monitoring the temperature. You might want to change the coordinates of probe 4.