

BEE 3500 Homework Assignment 5

Notes:

- For this assignment, you will use the computational software COMSOL Multiphysics 5.3, available in Academic Computing Center Engineering Library (ACCEL) at the Carpenter Hall. Rooms having access to COMSOL are the Blue Room and the Sunset Grill on the Ground Floor and the Green Room, Orange Room and Red Room on the 2nd floor. You can find when the facilities are open at www.accel.cornell.edu

- **Attached are the pages of a tutorial that you will complete. Instructions on what to submit is included in the following page.** Although the process will not be particularly time consuming, we encourage you to get started early since we can always experience software and hardware related issues.

- **For this week only, the TAs will be at the ACCEL facility, instead of their normal office hours.**

- We welcome you to try out other variations of the problem in the tutorial, or any other problem that you feel like. You will find this software tool to be quite flexible and user-friendly. Please do not turn in these other results with the homework assignment.

Thermal ablation of hepatic tumors

Go through the case study and submit the materials asked for. Each student needs to submit this individually. Color plots are not necessary.

What to submit:

- 1) Write down the specific Governing Equation that is being solved.
- 2) Provide the Boundary Conditions and Initial Conditions used.
- 3) List clearly all the input parameter values used in consistent units.

Follow the instructions given below and submit printouts for the following:

- 4) Create a movie of the heat conduction process. Follow these instructions to create a movie:
 - a. After running the problem, under **Model Builder** >> **Results**, right click on **Export**
 - b. Click **Animation** >> **File**.
 - c. Under **Settings** >> **Output** >> **Format**, select **AVI**.
 - d. Under **Filename** >> **Browse**. Input a name for the movie file and save it in desired folder.
 - e. Select the times at which you want to save the results for the movie: under **Animation Editing** >> **Time selection** >> **From list**. Hold Ctrl key and select 7 different times 0, 30, 60, 90, 120, 150 and 180 by left clicking. (Note that by default all times are selected. You do not want to use that setting as the computer will need a lot of memory to make the movie file.)
 - f. Click on **Export** button on top of the **Settings** panel or press F8.
 - g. The address of the produced file will be shown in the **Messages** window (the file is saved in the same folder as the COMSOL file)
 - h. Finally, make a contour plot of the temperature at $t = 180$ s. Expand **Isothermal Contours (ht)** >> **Contour**. Select **Study 1/Solution 1 (sol1)** as **Data set** in **Settings** window and select **Unit** to **degC**. Click on **Plot** button next.
 - i. Print and submit this contour plot. *You do not need to submit the movie file.*
- 5) Temperature as a function of time.
 - a. Plot the temperature at the point (0.0075, 0.01) as a function of time.
 - b. Save the data in text format (see instructions below)
 - c. Change the boundary condition. The probe temperature has been set at a temperature of 90°C. Change the temperature to 70°C.
 - d. Run the problem.
 - e. Save the data in text format.
 - f. Make one plot using the data obtained in steps (b) and (e) in Excel.
- 6) To save the data in text format:
 - a. Plot temperature as a function of time.

- b. Under **Model Builder** >> **Results**, expand **1D Plot Group** and right click on **Point Graph**.
- c. Click on **Add Plot Data to Export**.
- d. Under **Settings** >> **output** >> **Filename** >> **Browse**. Input a name for the text file and save it in desired folder.
- e. Click on **Export** button on top of the **Settings** panel or press F8.
- f. The address of the produced file will be shown in the **Messages** window (the file is saved in the same folder as the COMSOL file)

I Thermal ablation of hepatic tumors

Heated probes can be used to kill tumors in the liver as well as in other parts of the body. Such a procedure is preferable over open surgery because it is minimally invasive. However, thermal ablation must be planned accurately so that the majority of the tumor is heated to 50 °C for destruction, and there is minimum damage to the healthy tissue.

Problem formulation

In this case study, we are interested in finding the time it would take for the probe to heat and kill a tumor that reaches 0.75 cm from the surface of the liver. The first step is to simplify the physical problem. The schematic of the actual system is shown in Figure 6.1(a). We will create a very simplified model for this first case study. We assume that the entire heating probe is in contact with the tumor to be destroyed and the heating is only along the x-direction. The problem can therefore be solved in 1D. However, for better illustration of the process we use a 2D computational domain, as shown in Figure 6.1(b), with no variation in the y-direction.

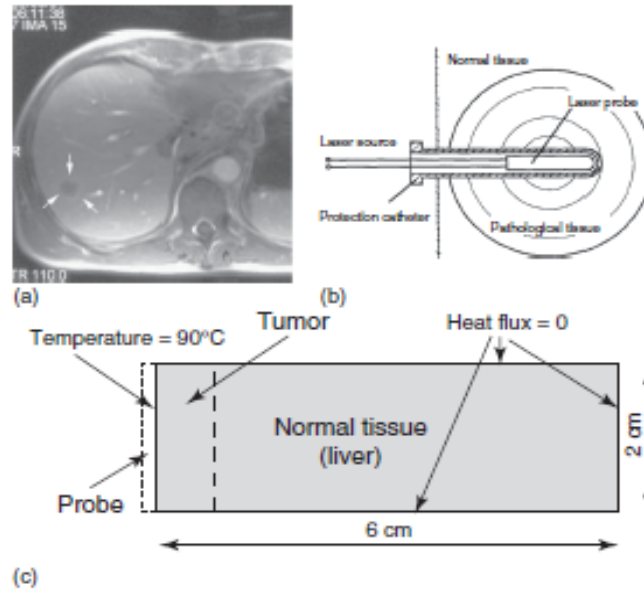
Governing equations Starting from the general heat transfer equation:

$$\underbrace{\frac{\partial T}{\partial t}}_{\text{transient}} + \underbrace{u \frac{\partial T}{\partial x}}_{\text{convection}} = \underbrace{\frac{k}{\rho C_p} \frac{\partial^2 T}{\partial x^2}}_{\text{conduction}} + \underbrace{\frac{Q}{\rho C_p}}_{\text{source}} \quad (6.1)$$

For our problem, the temperatures are dependent on time; there is no fluid flow and there are no source terms. The only mode of heat transfer is by conduction. So the problem is a transient conduction problem with no convection and heat source. Therefore, the governing equation that needs to be solved is:

$$\frac{\partial T}{\partial t} = \frac{k}{\rho C_p} \frac{\partial^2 T}{\partial x^2} \quad (6.2)$$

Notice here that the governing equation is written in 1D only. There is no variation in the other (y) direction.

**Figure 6.1**

(a) A liver tumor (between the arrows); (b) schematic of the process; (c) the computational domain showing the boundary conditions. Note that the probe is not included in the computational domain. Schematic reproduced with kind permission from Springer Science+Business Media: Ma *et al.* (2004).

Boundary conditions The boundary conditions are shown in Figure 6.1(b). To simplify the problem, the probe is assumed to be at a constant temperature of 90°C placed at the left edge of the tumor. The right edge of the normal tissue is considered to be far away and hence the heat flux is zero. As discussed earlier, there is no temperature variation in the y direction and therefore the boundary condition on the top and bottom surfaces are insulation (or heat flux set to zero).

Input parameters The density, thermal conductivity and specific heat are 1060 kgm^{-3} , 0.512 W (mK)^{-1} and 3600 J (kgK)^{-1} , respectively. The properties of the normal tissue and tumor are assumed to be the same.

Reference

Ma, N., X. Gao, and X. X. Zhang. (2004). Two-layer simulation model of laser-induced interstitial thermo-therapy. *Lasers in Medical Science*, **18**(4): 184–189.

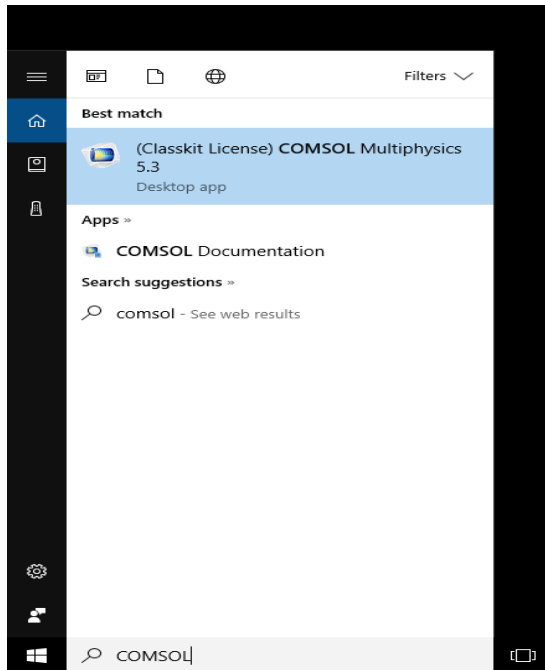
Implementation in COMSOL

Box 6.3**Steps**

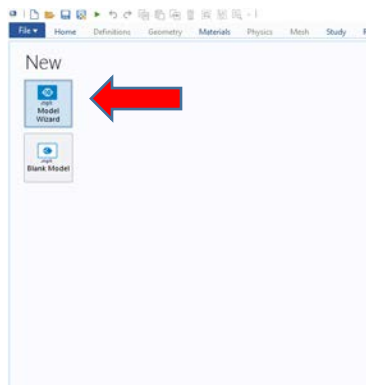
- (1) **Specifying the problem type** We will setup the problem as a transient heat transfer problem in 2D with no convection and heat source.
- (2) **Creating the geometry** We will draw a rectangle to model the normal tissue and tumor regions.
- (3) **Meshing** We will create a structured mesh in the domain.
- (4) **Defining material properties and initial conditions** We will specify the thermal conductivity, specific heat and density. We will also input the initial temperature.
- (5) **Defining boundary conditions** We will define the boundary conditions as shown in Figure 6.1(c).
- (6) **Specify solver parameters** We will solve the problem for 180 s.
- (7) **Postprocessing** We will plot the temperature history at a point. We will also plot the temperature distribution in the domain after 180s.
- (8) **Save and exit** We will finally save the file and exit.

SPECIFYING PROBLEM TYPE

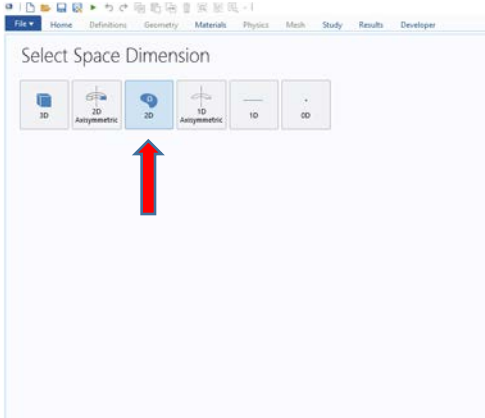
- 1) Search COMSOL in the search bar and select **(Classkit License) COMSOL Multiphysics 5.3** and save the file. Remember to save often to prevent losing work.



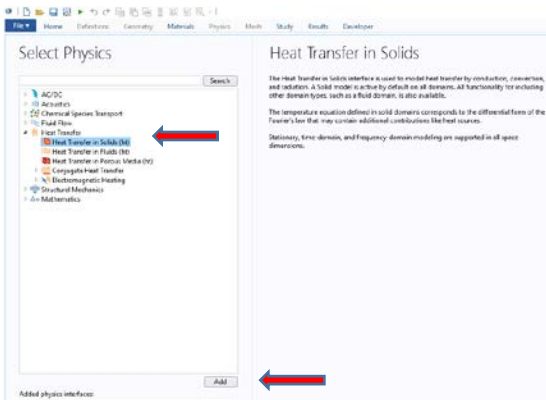
- 2) Select **Model Wizard**.



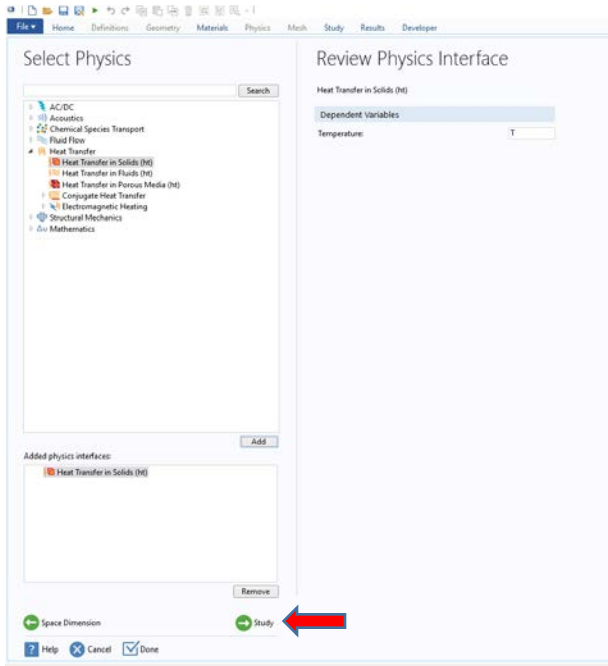
- 3) Select 2D under **Select Space Dimension**.



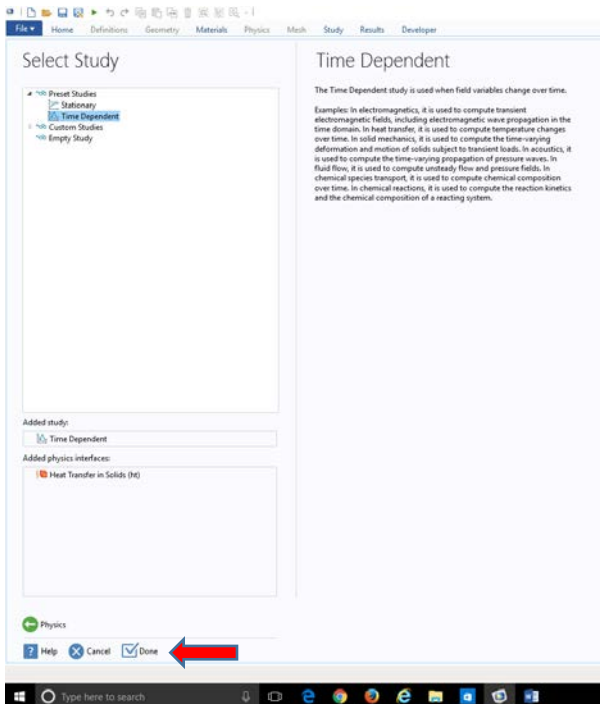
- 4) Under **Select Physics** click on the empty triangle left of **Heat Transfer** to expand the options >> **Heat Transfer in Solids (ht)**. More information about **Heat Transfer in Solids** should appear on the right side.



- 5) Click on **Add** button at the bottom of the Select Physics box. You should see **Heat Transfer in Solids (ht)** in the **Added physics interfaces:** box. Dependent variable T (Temperature) should appear in a box on right side under **Review Physics Interface**. Click on green arrow **Study** at the bottom of the window next.

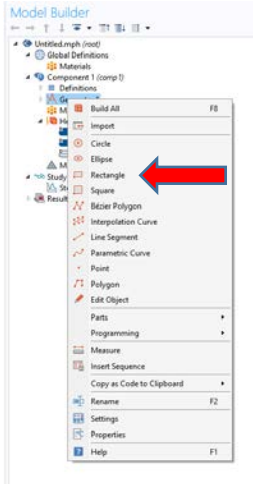


- 6) Select **Time Dependent** under select study. **Time Dependent** should appear in the **Added study:** box. Then click on the checked box **Done** at the bottom of the window.

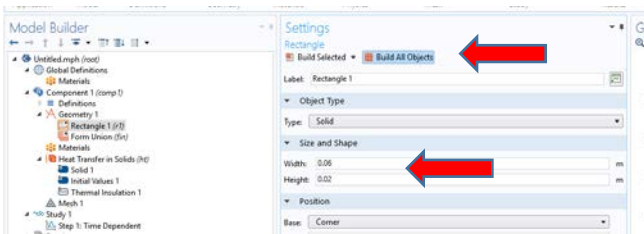


CREATING THE GEOMETRY

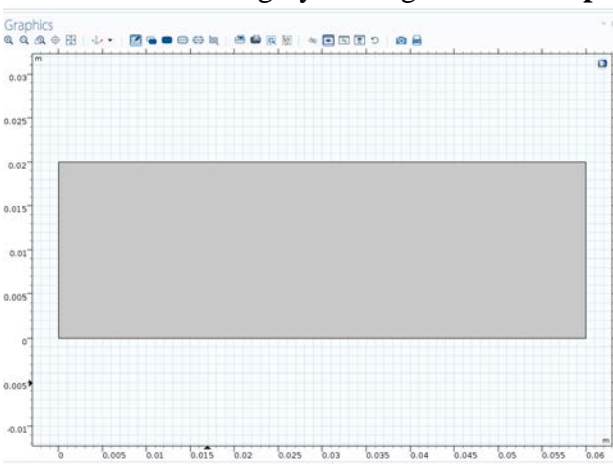
- 1) In the **Model Builder** window, right click on **Geometry 1**. Select **Rectangle** to add new geometry.



- 2) In the **Settings** window, set the **Width** of rectangle as 0.06m and **Height** as 0.02m in **Size and Shape**. Then click the orange **Build All Objects** icon.

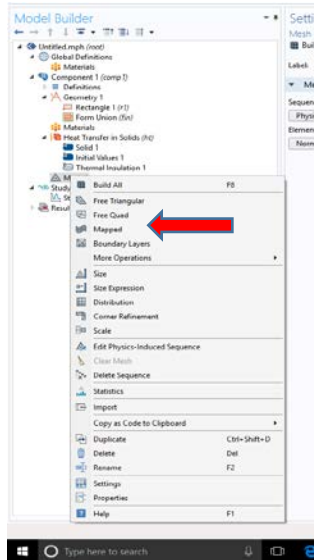


- 3) It should result in a grey rectangle in the **Graphics** window on the right side.

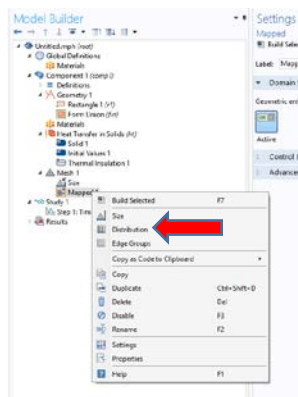


MESHING

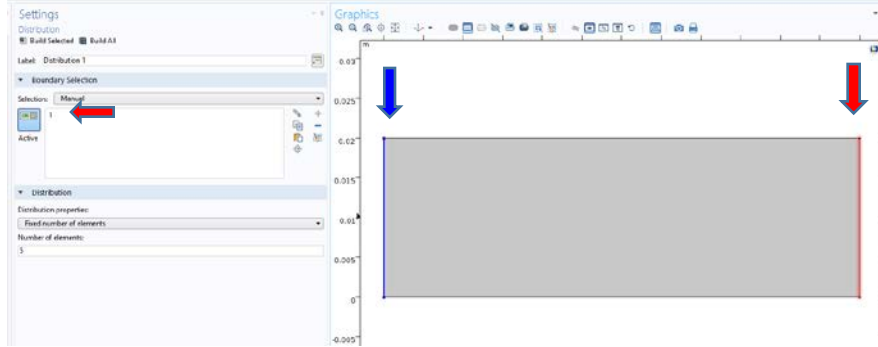
- 1) Under **Model Builder** right click on **Mesh 1** and select **Mapped**.



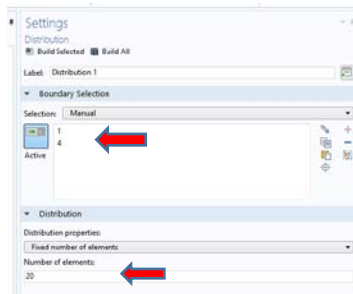
- 2) **Mesh 1** option should expand and, two options - **Size** and **Mapped 1** should appear underneath. Right click on **Mapped 1** and select **Distribution** option.



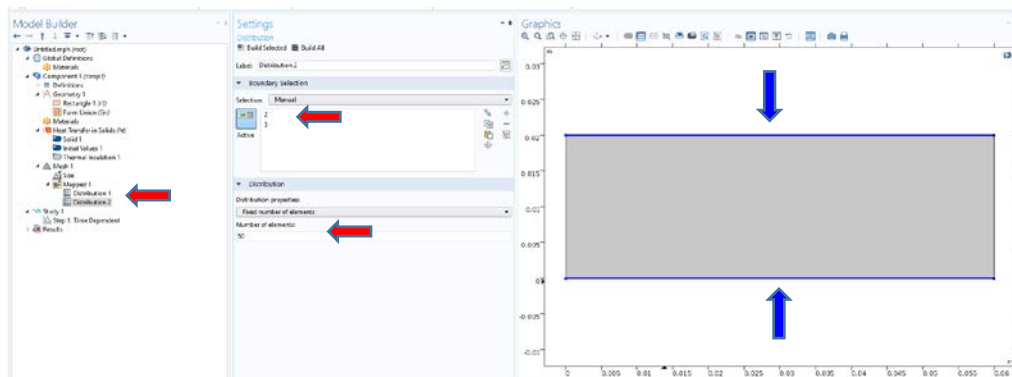
- 3) Repeat step 2) to create two distributions- **Distribution 1** and **Distribution 2**. Click on **Distribution 1**. Under **Graphics** window, click on boundary 1 and 4 (left and right boundaries of the grey rectangle). Edge of the rectangle will turn **red** if cursor hovers over it and turn **blue** after the selection.



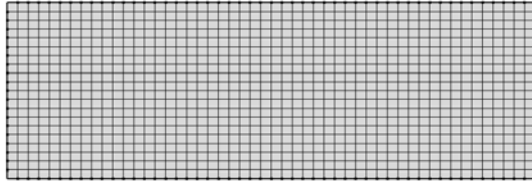
- 4) Input **Number of elements:** as 20 under **Distribution** in **Settings** window. Make sure **1** and **4** appear in the **Boundary Selection** under **Settings** window.



- 5) Select **Distribution 2**. Select boundary 2 and 3 (top and bottom boundaries) in the **Graphics** window. Make sure **2** and **3** appear in the **Boundary Selection** under **Settings** window. Input **Number of elements:** as 50 under **Distribution** in **Settings** window.

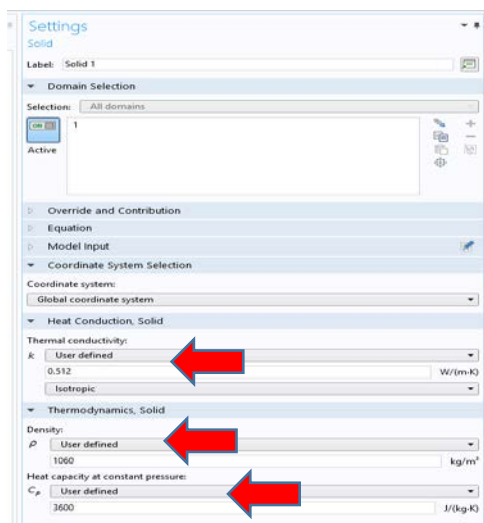


- 6) Click on the dark blue **Build All** button to create a mapped mesh.

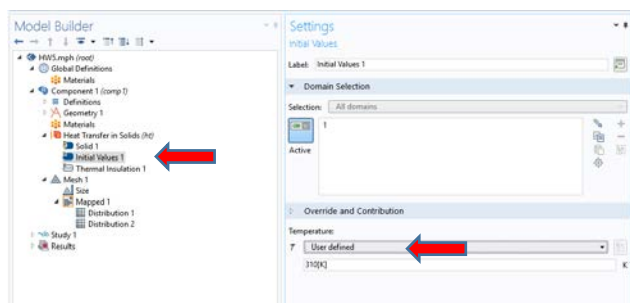


DEFINING MATERIAL PROPERTIES AND PARAMETERS

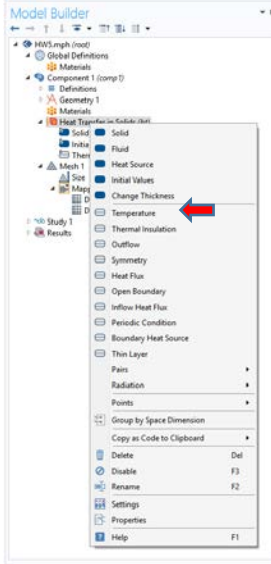
- 1) In the **Model Builder** expand **Heat Transfer in Solids (ht)** and click on **Solid 1**.
- 2) Under **Heat Conduction, Solid**, select **User defined** for **Thermal conductivity (k)** and input value 0.512 W/m·K. Similarly, select **User defined** for **Density (ρ)** and **Heat capacity at constant pressure (C_p)** under **Thermodynamics, Solid** and input values 1060 kg/m³ and 3600 J/kg·K respectively.



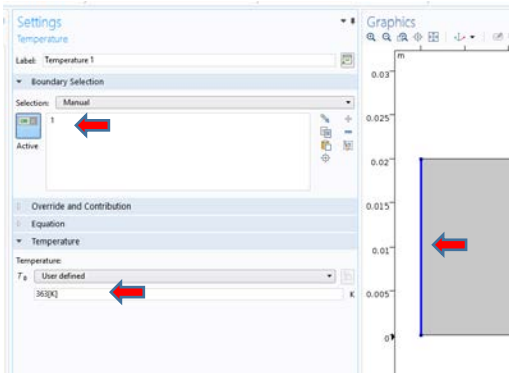
- 3) In the **Model Builder** expand **Heat Transfer in Solids (ht)** and click on **Initial Values 1**. Select **User defined** under **Override and Contribution**. Input value 310 K.



- 4) Right click on **Heat Transfer in Solids (ht)** in the **Model Builder**. Select **Temperature** to add new constant temperature boundary condition.

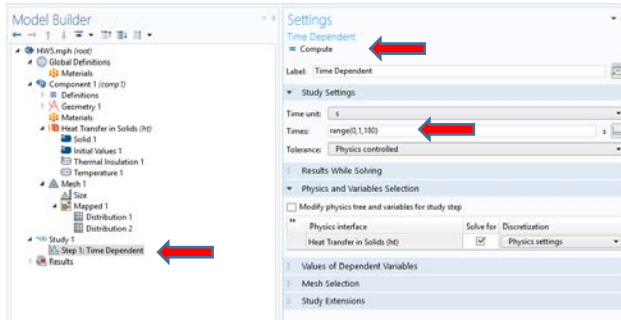


5) Under **Graphics**, select boundary 1 (left edge) and set boundary 1 temperature to 363 K.

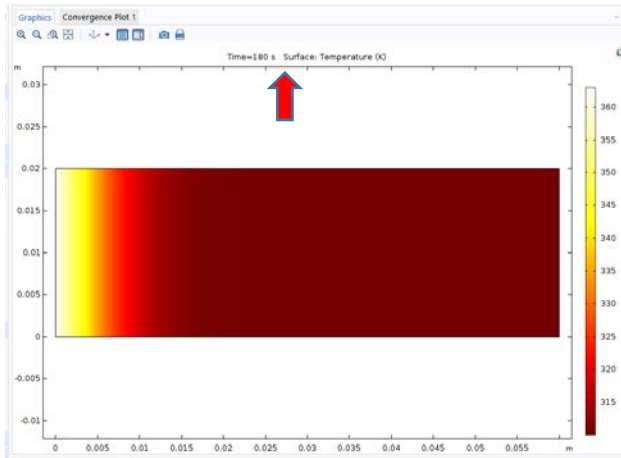


COMPUTATION AND ANALYSIS

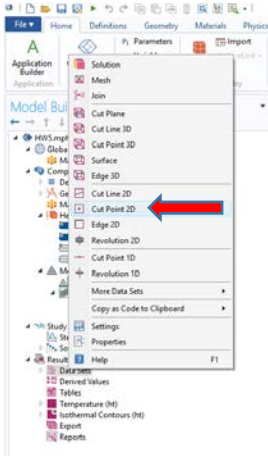
- 1) Expand **Study 1** in the **Model Builder** and select **Step 1: Time Dependent**.
- 2) In the **Setting** window, change the **Times** under **Study Settings** to **range(0, 1, 180)**. It means the simulation will run from 0 seconds to 180 seconds for time interval of 1 second. Click the icon with two blue horizontal lines for **Compute** next.



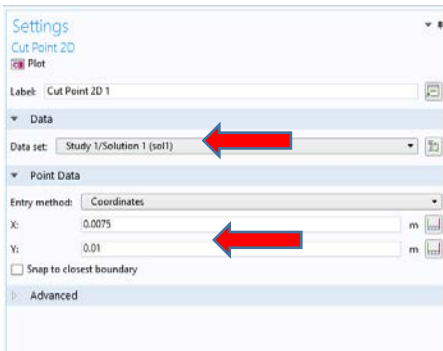
- 3) It should result in surface map at 180 seconds in the **Graphics** window.



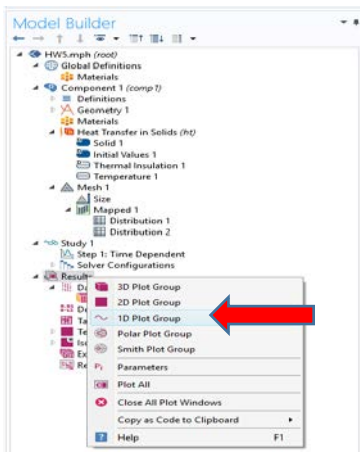
- 4) Right click on **Data Sets** under **Results** in the **Model Builder** and select option **Cut Point 2D**.



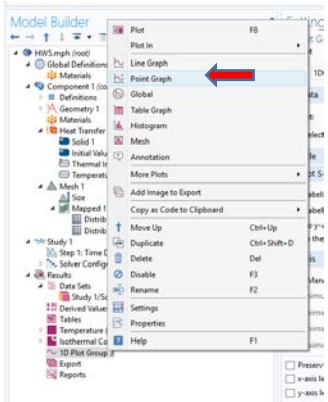
- 5) In the **Settings** window, select **Study 1/Solution 1 (sol1)** under pulldown option for **Data set**. Input 0.0075 m for **X** coordinate and 0.01 m for **Y** coordinate.



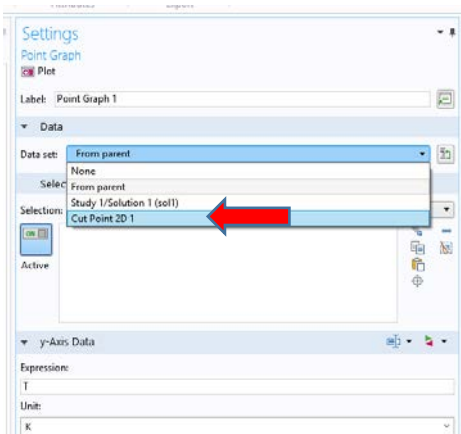
- 6) Under **Model Builder** window, right click on **Results** and select **1D Plot Group**.



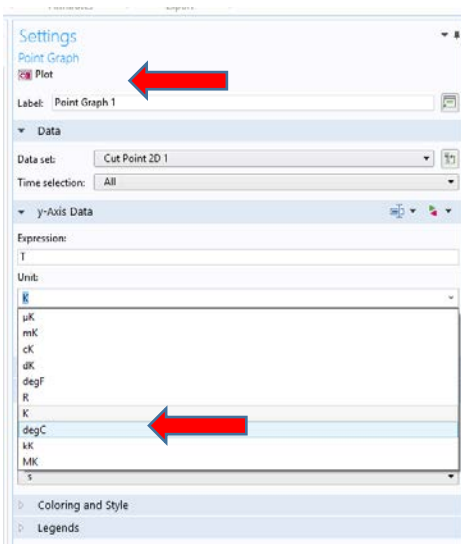
- 7) Right click on **1D Plot Group 3** and select **Point Graph**.



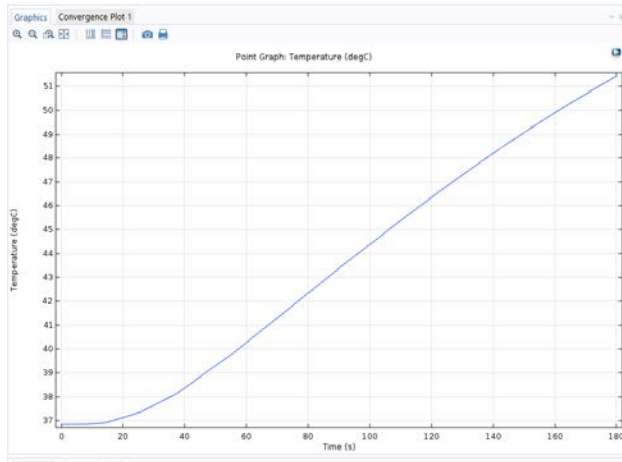
8) In the **Settings** window, select **Cut Point 2D 1** from pulldown options for **Data set**.



9) Under **y-Axis Data**, change the **Unit** to **degC** from dropdown menu and click the **plot** button.

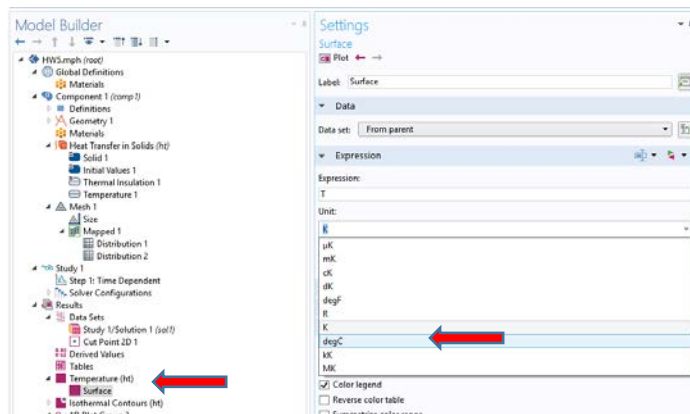


10) You should get temperature profile at (0.0075, 0.01) in the **Graphics** window.

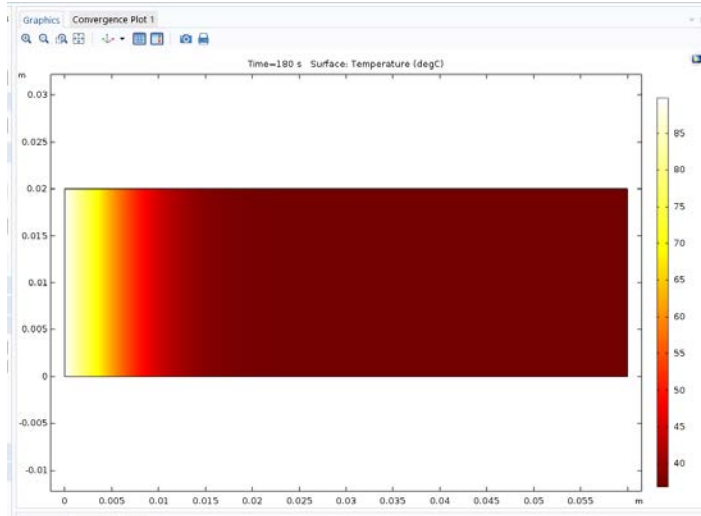


11) Expand **Temperature (ht)** under **Results** in the **Model Builder** and select **Surface**.

12) In the **Settings** window, select **degC** from the dropdown menu for **Unit** under **Expression**.



13) Click on the folder icon with arrow for **plot**. You should get final surface plot in degree Celsius.



NOTE: You can always return to the surface plot by clicking on **Temperature (ht)** in the **Model Builder**.