

RADIOFREQUENCY CARDIAC ABLATION

Modelling and simulation using COMSOL Multiphysics

OBJECTIVES

- Solving biomedical transport problems using COMSOL Multiphysics
- Simulating radiofrequency ablation for treating cardiac arrhythmias

WHAT IS ARYTHMIA

- Irregular beating of the heart
- Causes decreased blood flow and oxygen supply to the brain and body

RADIOFREQUENCY ABLATION(RFA)

- Ablation is a process of removing body tissues
- RFA –
 - Destroys tumors that cannot be removed surgically
 - Through controlled heating
 - Minimally invasive

PROBLEM FORMULATION

- A cylindrical electrode is introduced into the middle of the selected myocardial tissue.
- Tissue properties are homogenous.
- Joule heat generation by resistive heating.
- Tissue temperature must go more than 50°C.

GOVERNING EQUATIONS

- Penne's bioheat transfer equation:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \cdot \nabla T) = \rho_b C_b v_b (T_b - T) + Q$$

Here, $\rho_b C_b v_b (T_b - T)$ is the blood perfusion term.

Where,

ρ_b = Blood Density

C_b = Specific Heat capacity of Blood

v_b = Blood perfusion Rate

T_b = Arterial Blood Temperature

T = Local tissue temperature

- $Q = \sigma |\nabla V|^2$ is from the electrode

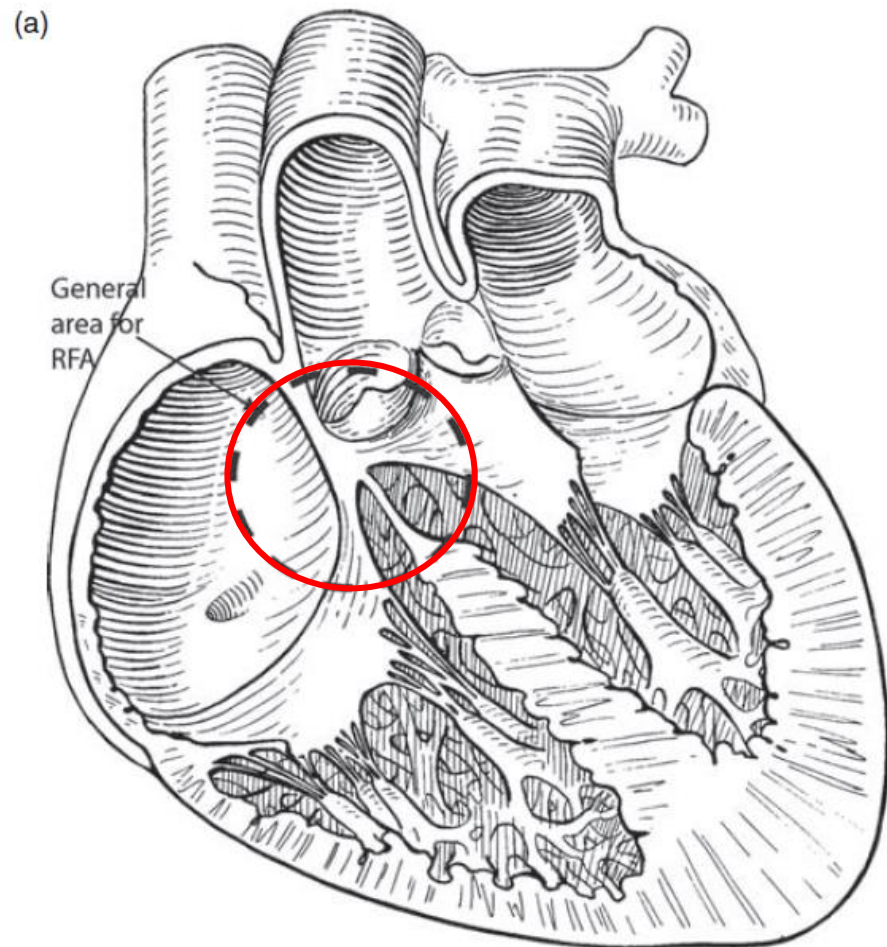
Where,

σ = Electrical conductivity

V = Electrical potential

GEOMETRY

A



B

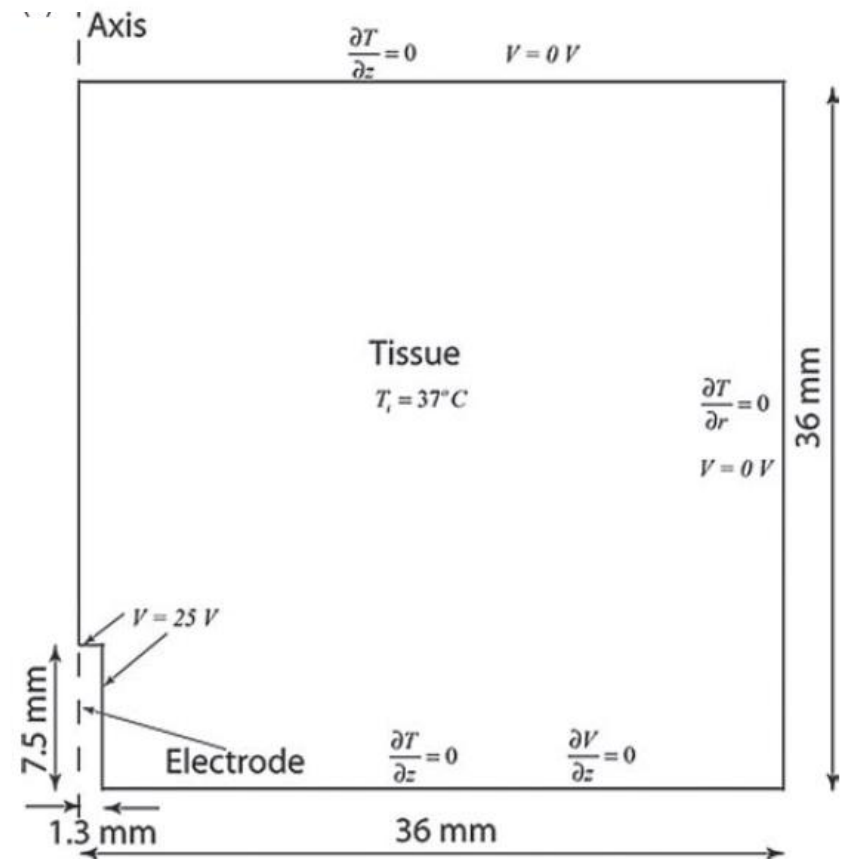
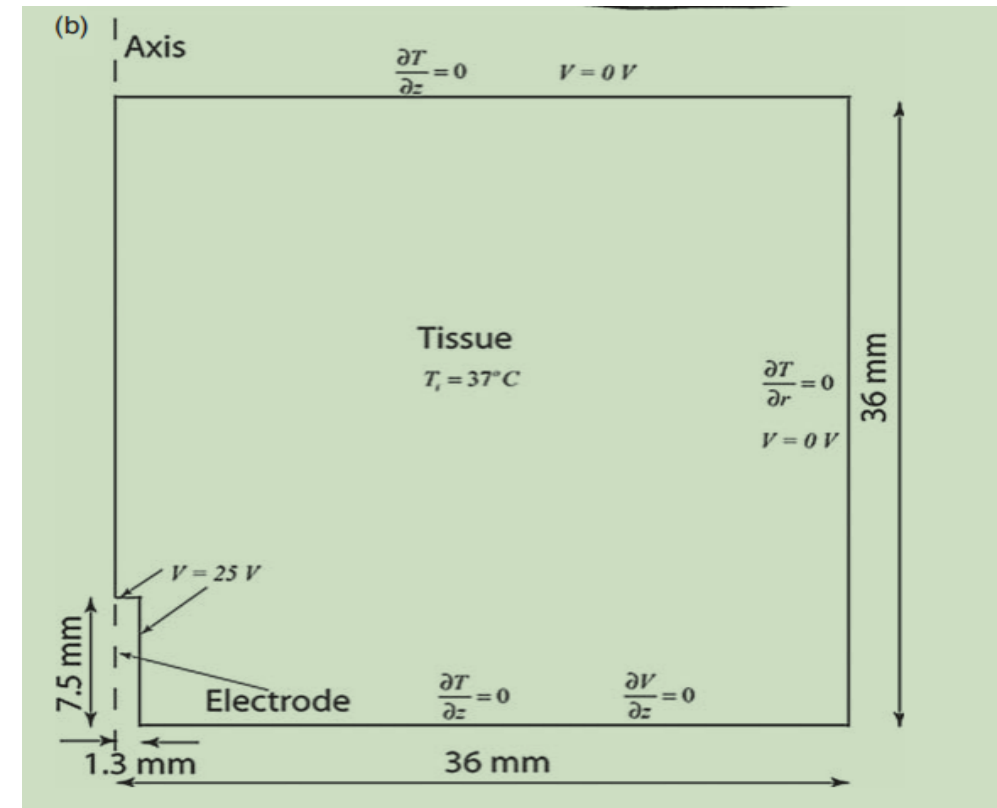


Figure 6.14

BOUNDARY CONDITIONS

- Axisymmetric
- Left surface is the axis
- Initial Temperature- 37°C
- Electrode is excluded in the geometry
- Electric potential to the surface of electrode is implemented as a boundary condition
- Heat fluxes at all surface are zero



INPUT PARAMETERS

Parameter	Value
Thermal conductivity of the tissue, k	$0.4925 + 0.001\ 195T\ \text{Wm}^{-1}\text{K}^{-1}$
Specific heat of the tissue, C_p	$3200\ \text{Jkg}^{-1}\text{K}^{-1}$
Density of the tissue, ρ	$1200\ \text{kgm}^{-3}$
Duration of heating, t	60 s
Blood perfusion coefficient, $\rho_b C_{p,b} \dot{V}_b^v$	$2000\ \text{Wm}^{-3}\text{K}^{-1}$
Electrical conductivity, σ	$0.222\ \text{S m}^{-1}$
Arterial blood temperature, T_a	37 °C
Initial tissue temperature, T_i	37 °C
Electric potential at the electrode surface, V	25 V

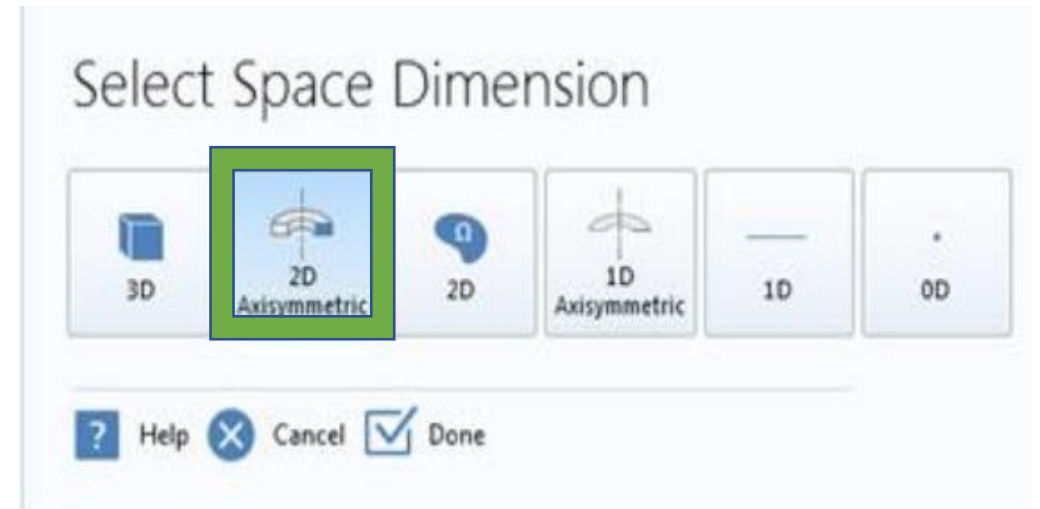
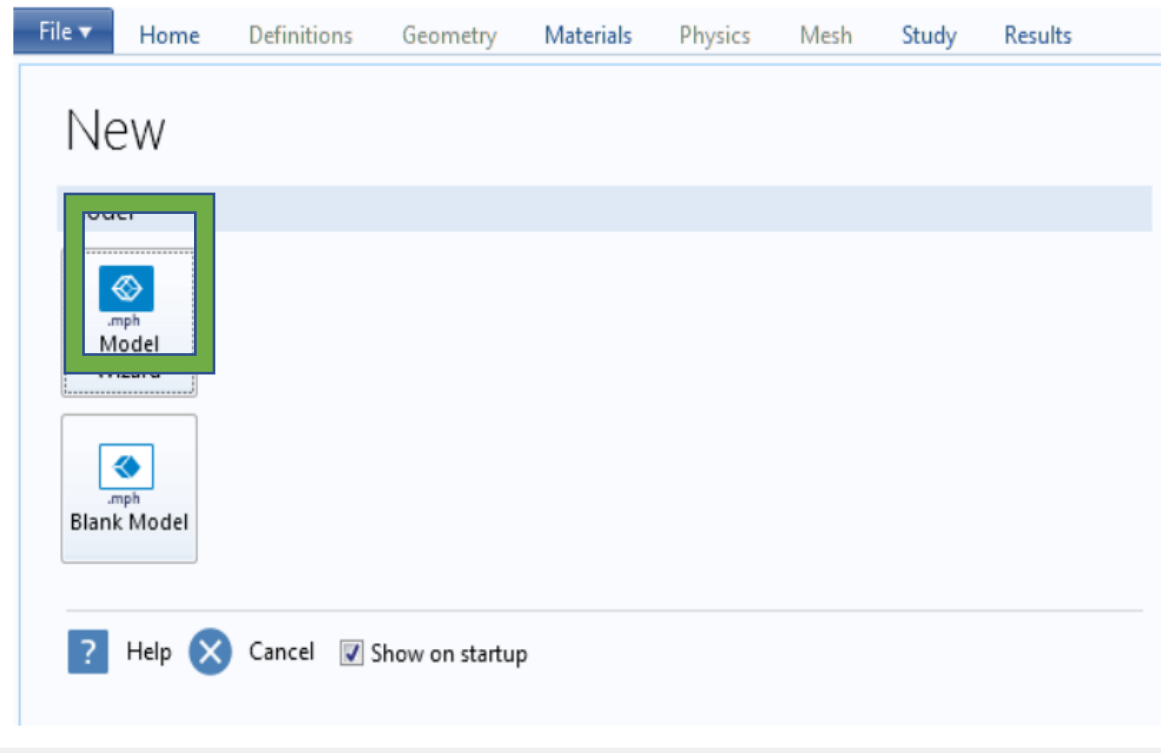
DEFINITION OF EXPRESSION

- Two most significant expression:
- k – thermal conductivity of tissue: $0.4225+0.001195*T$
- Q_{blood} – Blood perfusion heat
- Q_{dc} – heat source term due to radiofrequency (not used in Comsol Multiphysics v5.4/v5.5)
- Operational time: 60s

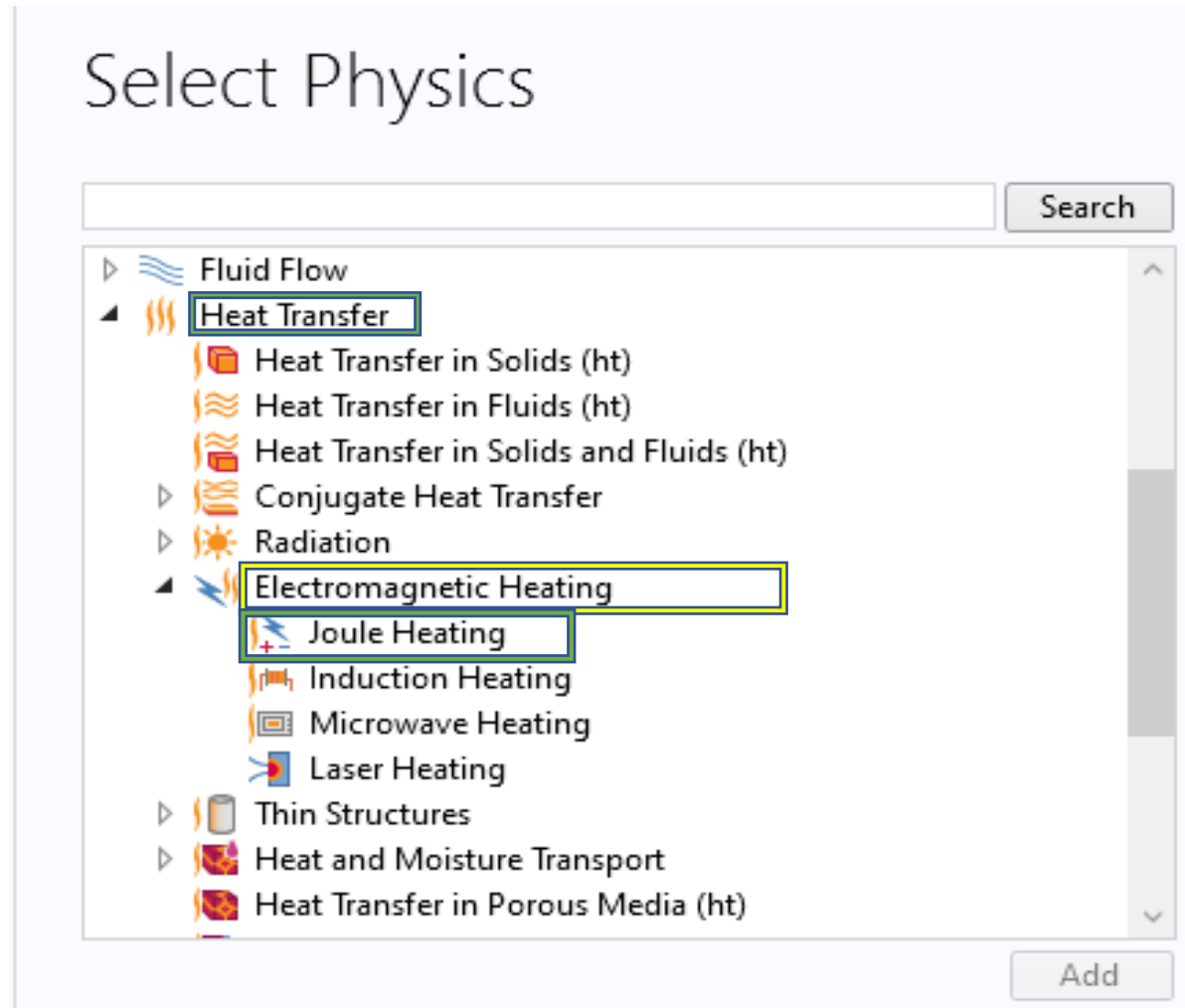
SIMULATION STEPS

- Problem type specification:

Model wizard  **space dimension**  **2D Axisymmetric**



- Under **Select Physics** → **Heat transfer** → **Electromagnetic** → **Joule- heating**



- Under **Select Study** ➡ **time dependent**

Select Study

- General Studies
 - Stationary
 - Time Dependent**
- Preset Studies for Selected Physics Interfaces
 - Heat Transfer in Solids
 - Electric Currents
- Preset Studies for Selected Multiphysics
 - Frequency-Stationary
 - Frequency-Transient
 - Sequential Frequency-Stationary
 - Sequential Frequency-Transient

Added study:

Added physics interfaces:

- Electric Currents (ec)
- Heat Transfer in Solids (ht)
- Multiphysics
 - Electromagnetic Heating (emh1)

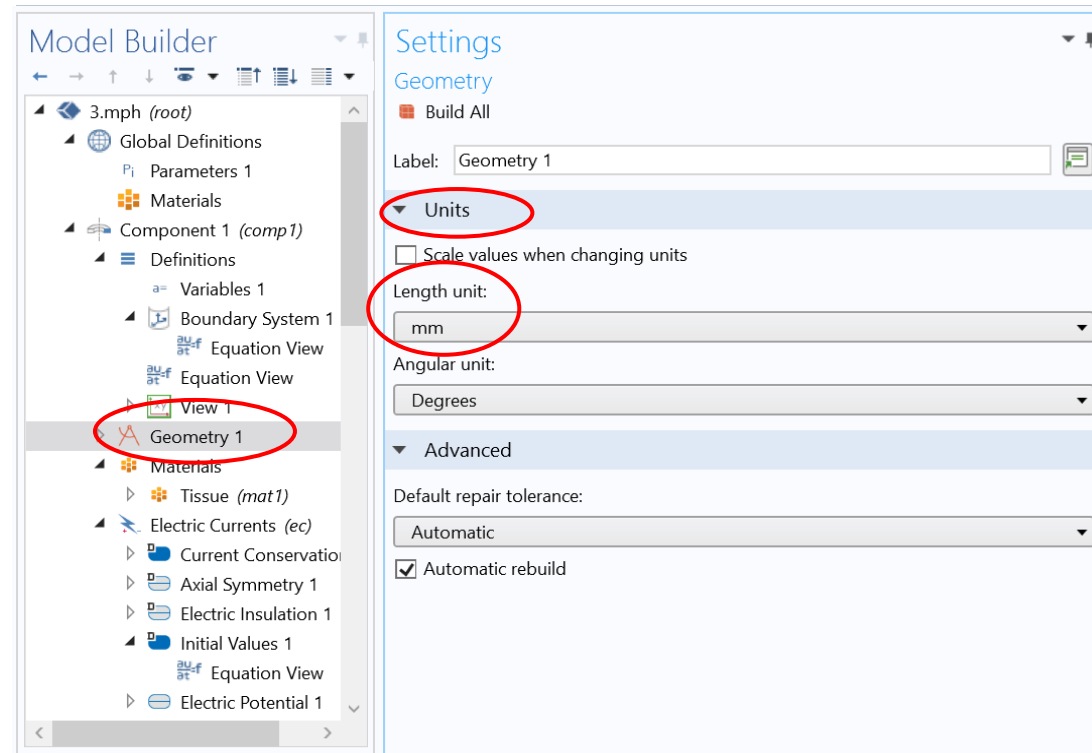
Physics

Help Cancel Done

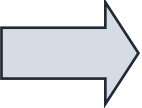
GEOMETRY CREATION

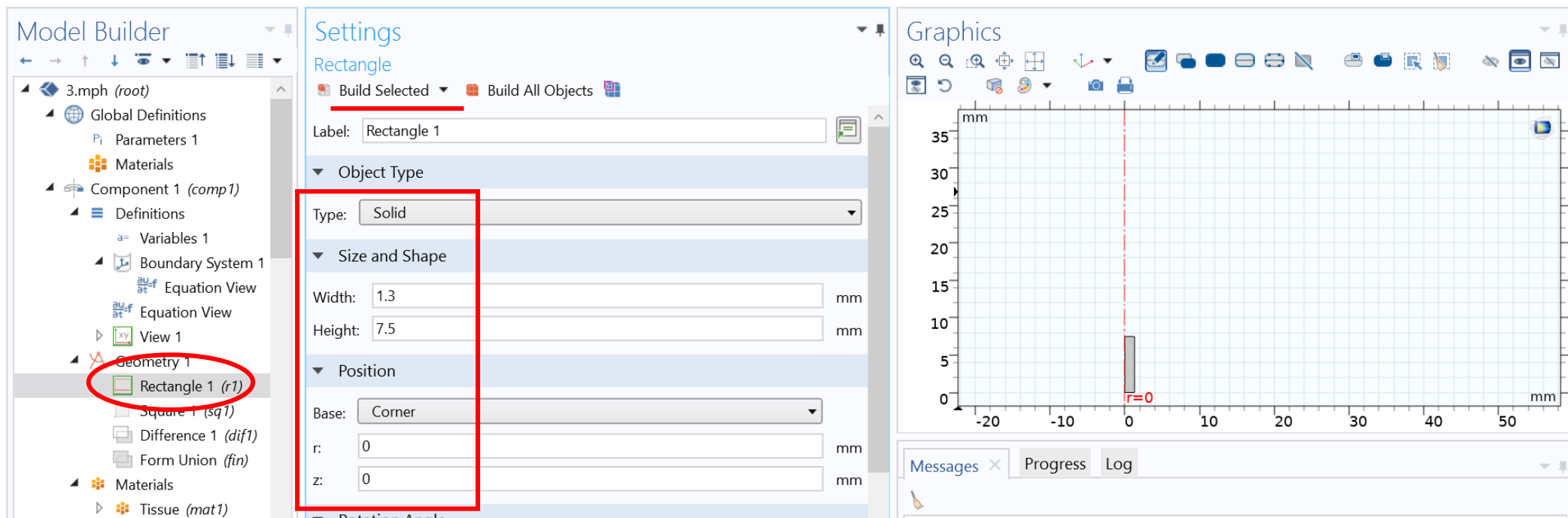
- Under **Model Builder**:

Geometry  **Units**



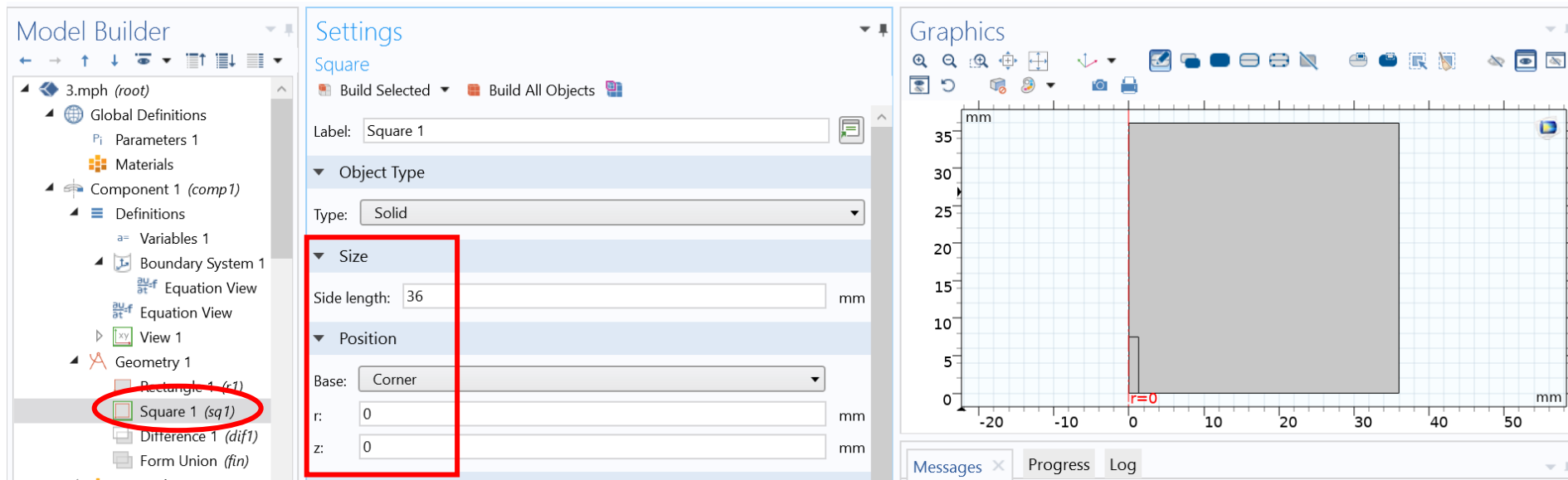
GEOMETRY CREATION

- Geometry  "Rectangle"



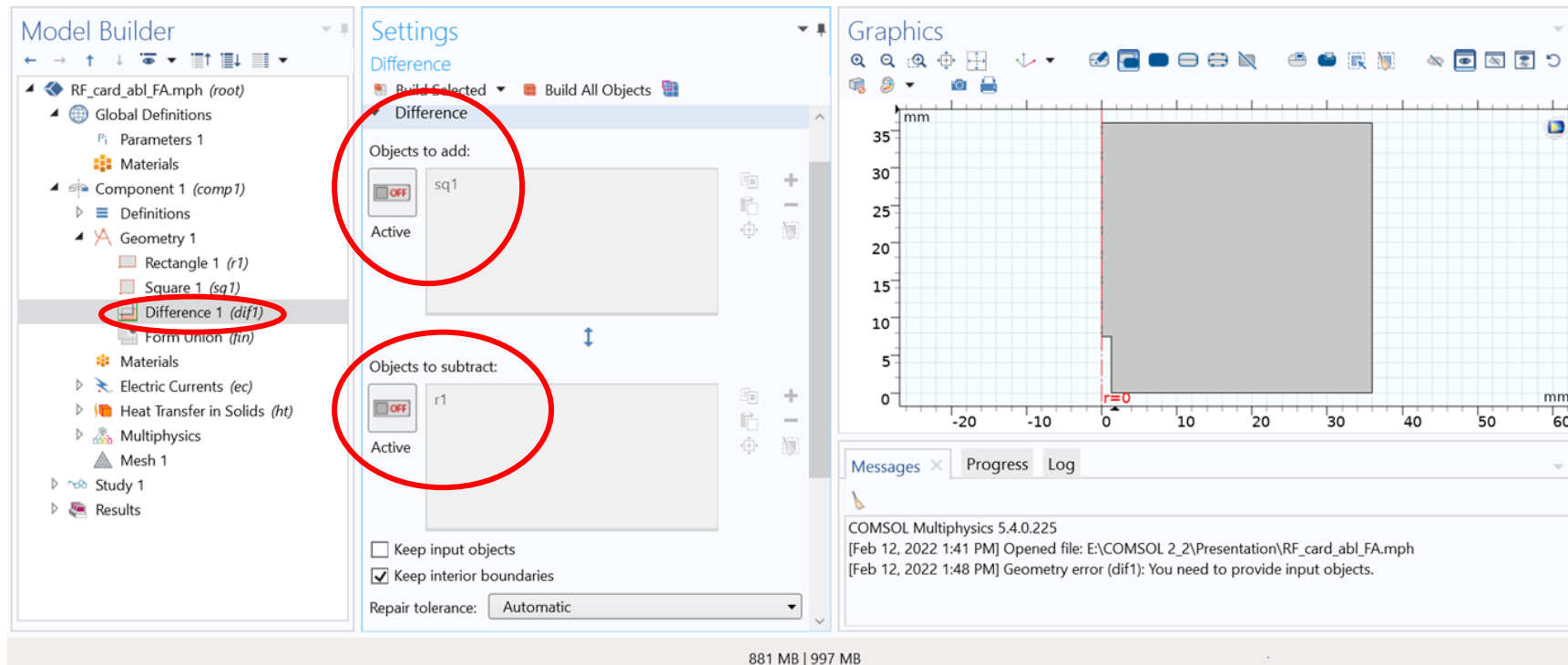
GEOMETRY CREATION

- Geometry → “Square”



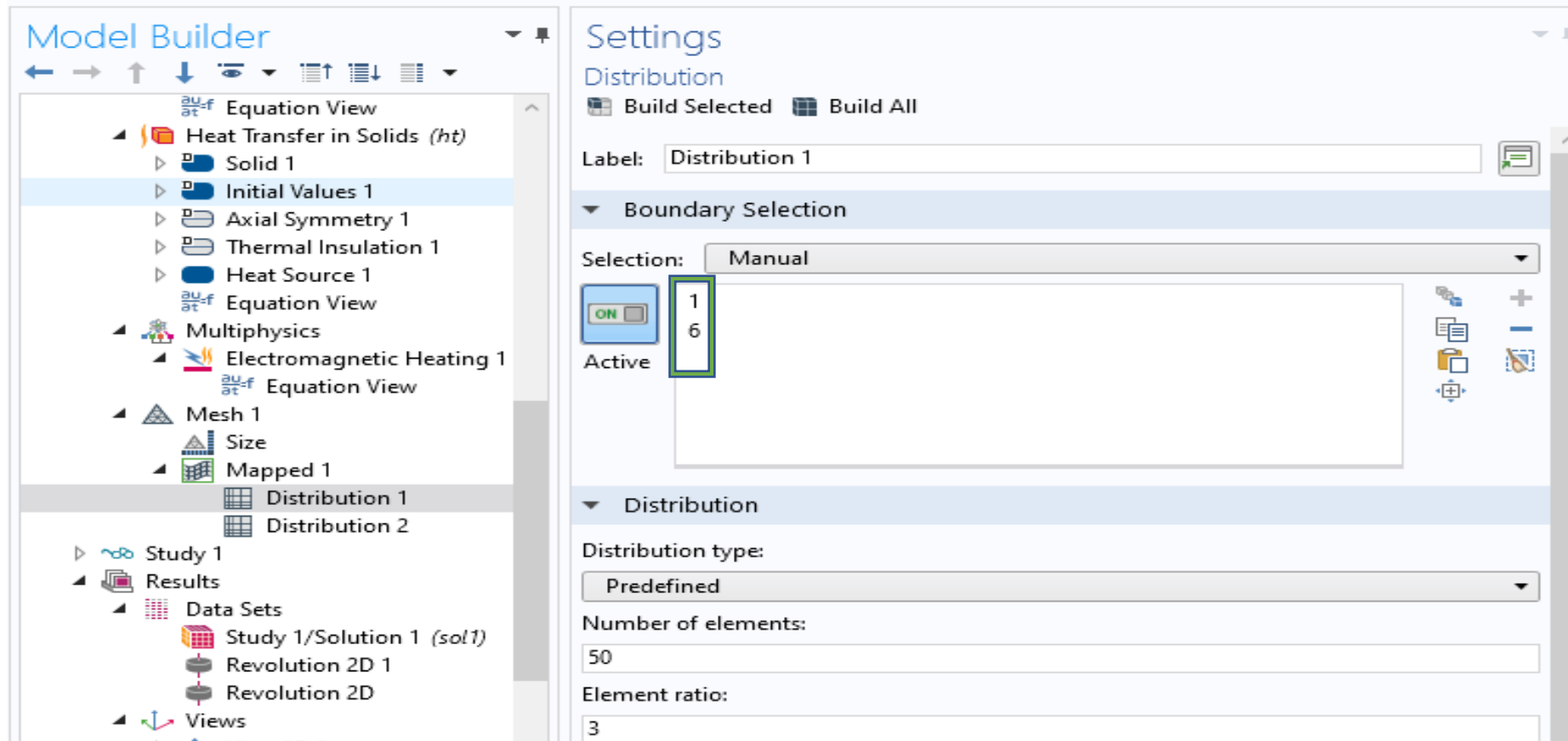
GEOMETRY CREATION

- **Geometry** ➡ **Booleans and Partitions** ➡ **Difference**

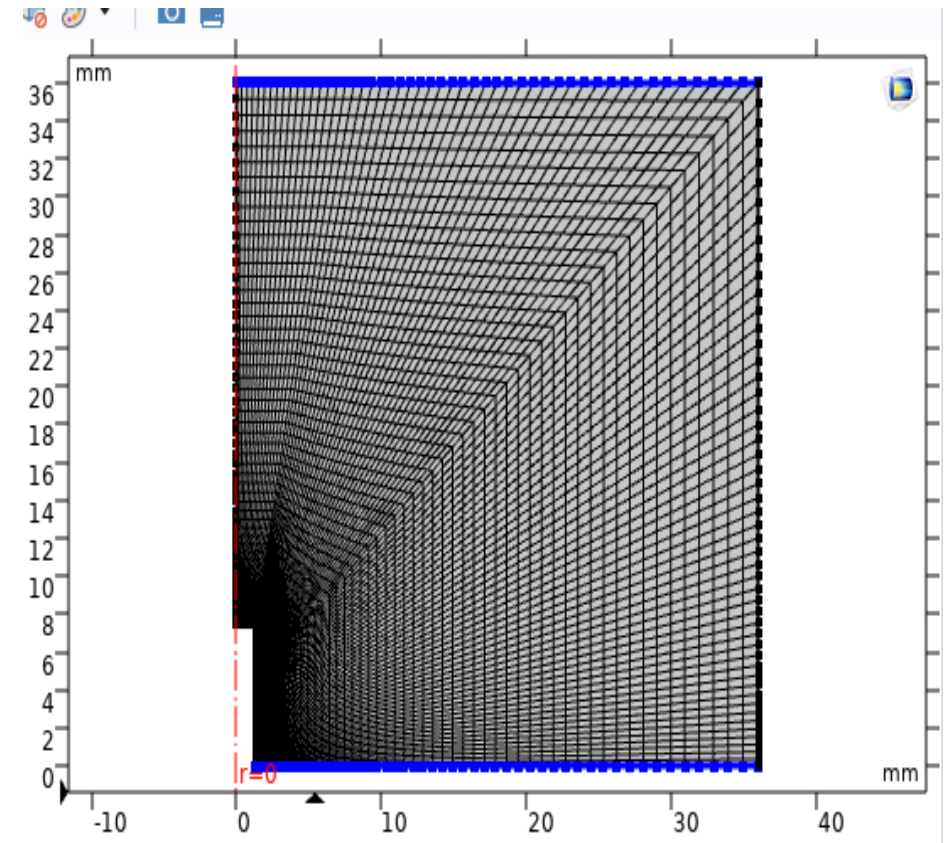
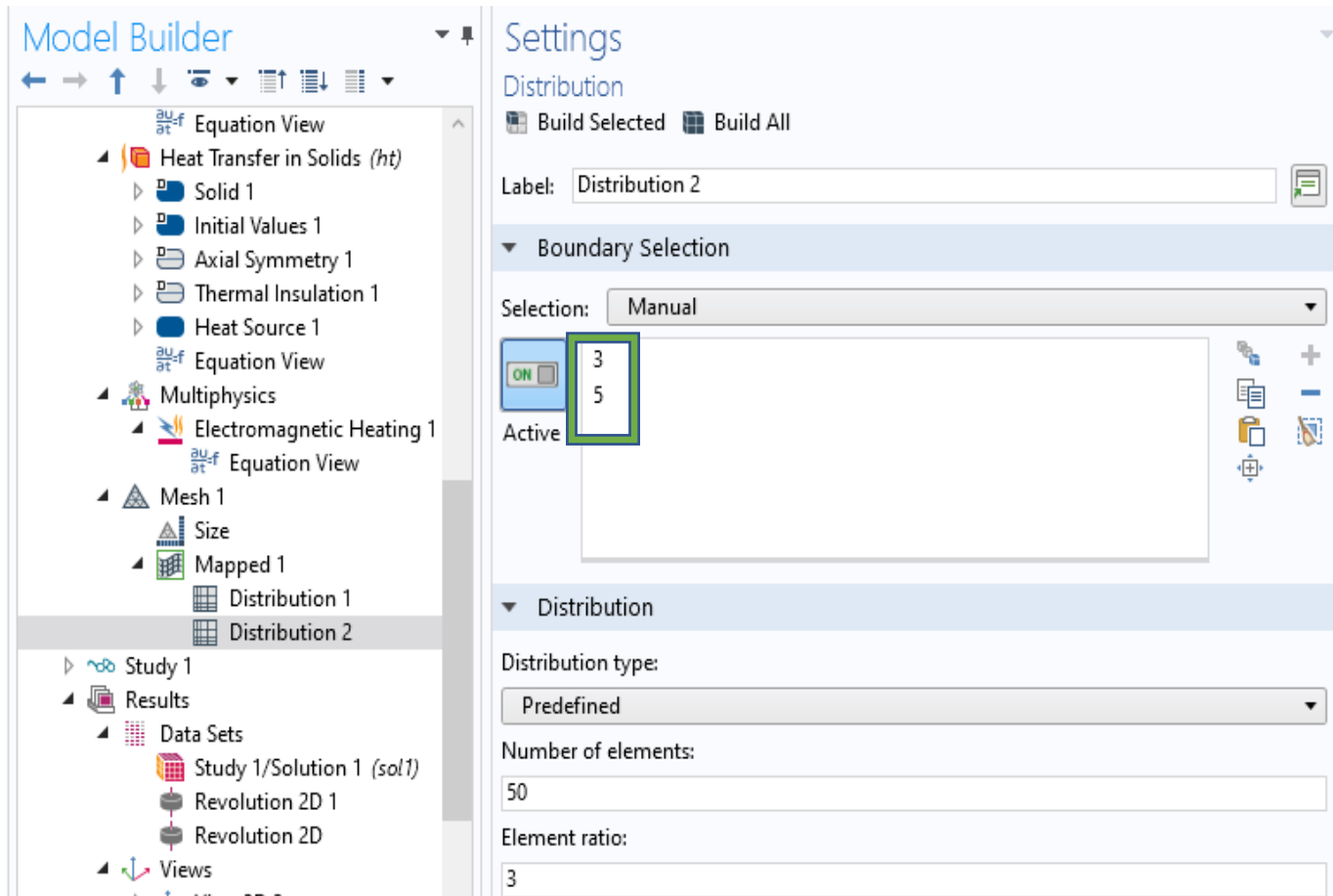


MESHING

- Mesh  Mapped  Distribution



MESHING



VARIABLE DECLARATION

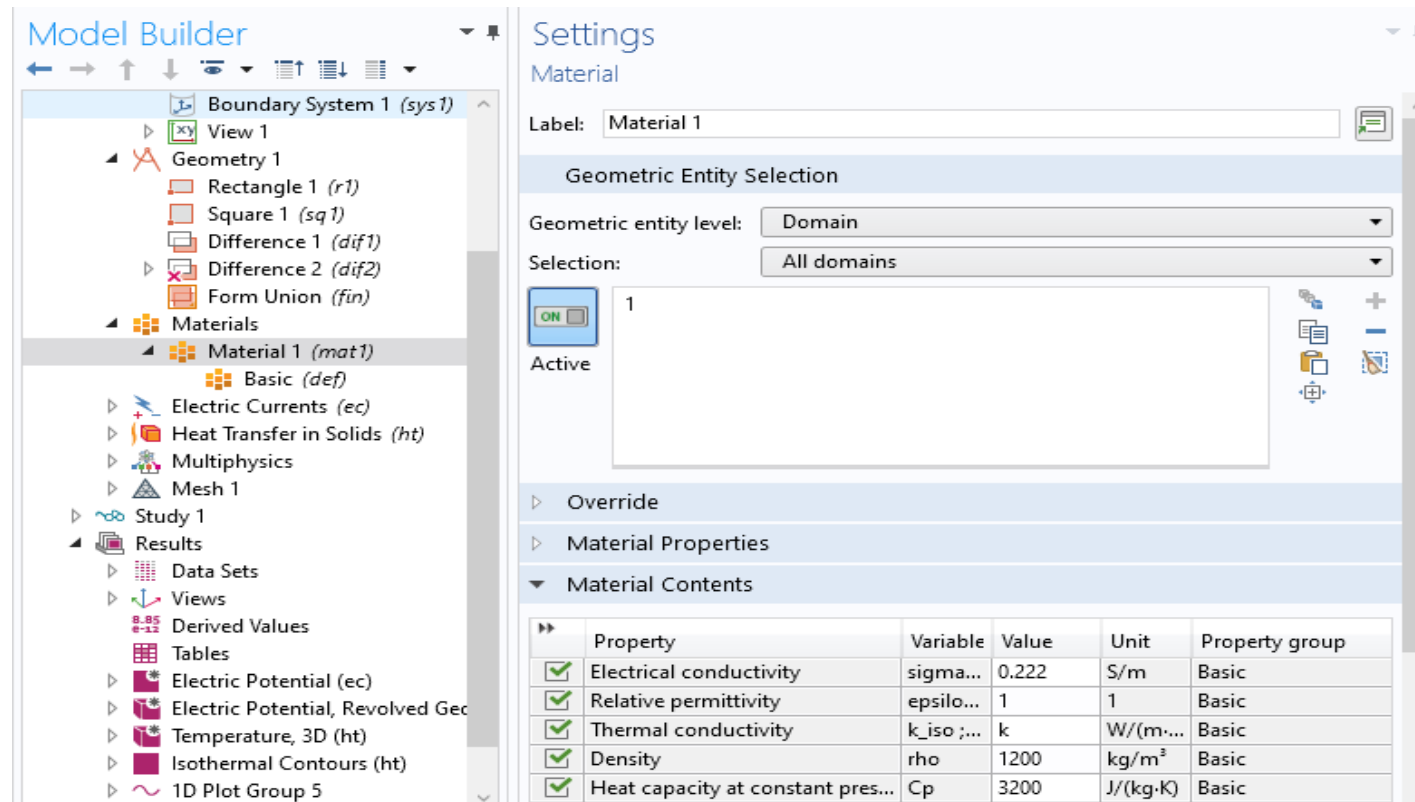
- K and Q_blood are set as variables.

The screenshot displays the COMSOL Multiphysics Model Builder interface. On the left, the 'Model Builder' tree shows the hierarchy: 3.mph (root) > Global Definitions > Parameters 1 > Materials > Component 1 (comp 1) > Definitions > Variables 1. The 'Variables' node is selected. On the right, the 'Settings' pane for 'Variables 1' is shown. It includes a 'Label' field set to 'Variables 1', a 'Geometric Entity Selection' section with 'Geometric entity level' set to 'Entire model', and an 'Active' checkbox that is checked. Below these, a table lists the declared variables.

Name	Expression	Unit	Description
k	$a \cdot (0.4925 + 0.001195 \cdot T)$	W/(m·K)	
Q_blood	$b \cdot (2000 \cdot (310.15 - T))$	W/m ³	
a	$1 [W/(K^2 \cdot m)]$	kg·m/(s...	
b	$1 [W/(K \cdot m^3)]$	W/(m ³ ·K)	

MATERIALS PROPERTY

Basic properties of tissue are defined under **Materials**

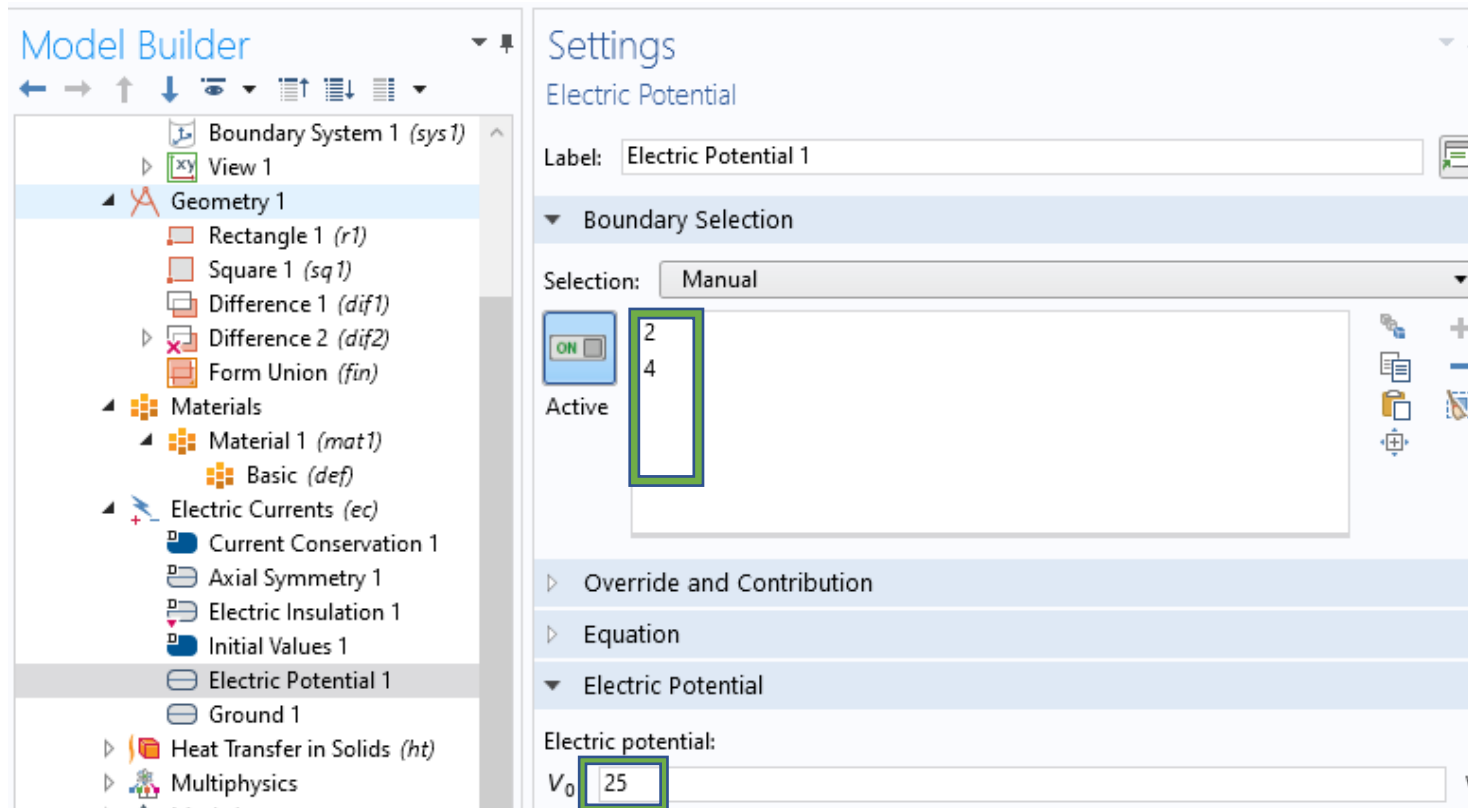


The screenshot displays the COMSOL Multiphysics interface. On the left, the **Model Builder** tree shows the hierarchy: **Boundary System 1 (sys1)** > **View 1** > **Geometry 1** > **Materials** > **Material 1 (mat1)**. The **Material 1 (mat1)** node is selected, showing its sub-items: **Basic (def)**, **Electric Currents (ec)**, **Heat Transfer in Solids (ht)**, **Multiphysics**, and **Mesh 1**. On the right, the **Settings** panel for **Material 1** is shown. It includes a **Label** field set to "Material 1", a **Geometric Entity Selection** section with **Geometric entity level** set to "Domain" and **Selection** set to "All domains", and a **Material Contents** table. The **Material Contents** table lists the following properties:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma...	0.222	S/m	Basic
Relative permittivity	epsilo...	1	1	Basic
Thermal conductivity	k_iso ;...	k	W/(m·...	Basic
Density	rho	1200	kg/m³	Basic
Heat capacity at constant pres...	Cp	3200	J/(kg·K)	Basic

BOUNDARY CONDITIONS

- Boundary conditions are given of Electric Potential, Electric insulation and Ground under **Electric current**



BOUNDARY CONDITIONS

The screenshot displays the COMSOL Multiphysics Model Builder interface. On the left, the 'Model Builder' tree shows a hierarchy starting with 'Boundary System 1 (sys1)', followed by 'View 1', 'Geometry 1', and 'Materials'. Under 'Materials', 'Material 1 (mat1)' is expanded to show 'Basic (def)'. Under 'Electric Currents (ec)', several physics are listed, with 'Ground 1' selected at the bottom.

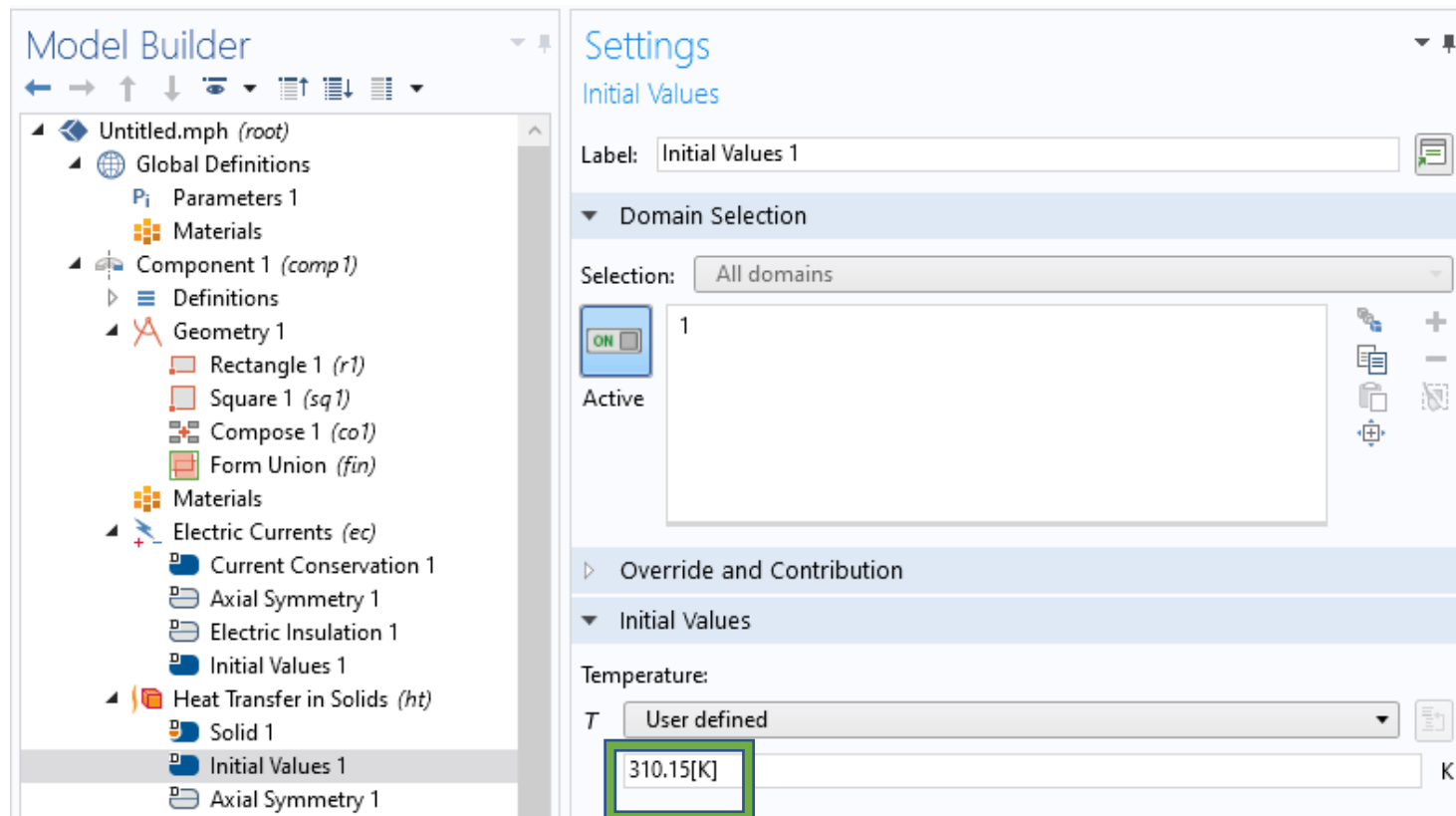
The central 'Settings' pane for 'Ground' shows the 'Label' as 'Ground 1'. Under the 'Boundary Selection' section, the 'Selection' is set to 'Manual'. A list of selected boundaries is shown, with '3' and '6' highlighted by a green box. The 'Active' checkbox is checked.

The right 'Settings' pane for 'Electric Insulation' shows the 'Label' as 'Electric Insulation 2'. Under 'Boundary Selection', the 'Selection' is set to 'Manual'. A list of selected boundaries is shown, with '5' highlighted by a green box. The 'Active' checkbox is checked.

Below the 'Boundary Selection' section, the 'Override and Contribution' section is expanded, showing the 'Equation' section. The 'Show equation assuming:' dropdown is set to 'Study 1, Time Dependent'. The equation $\mathbf{n} \cdot \mathbf{J} = 0$ is displayed.

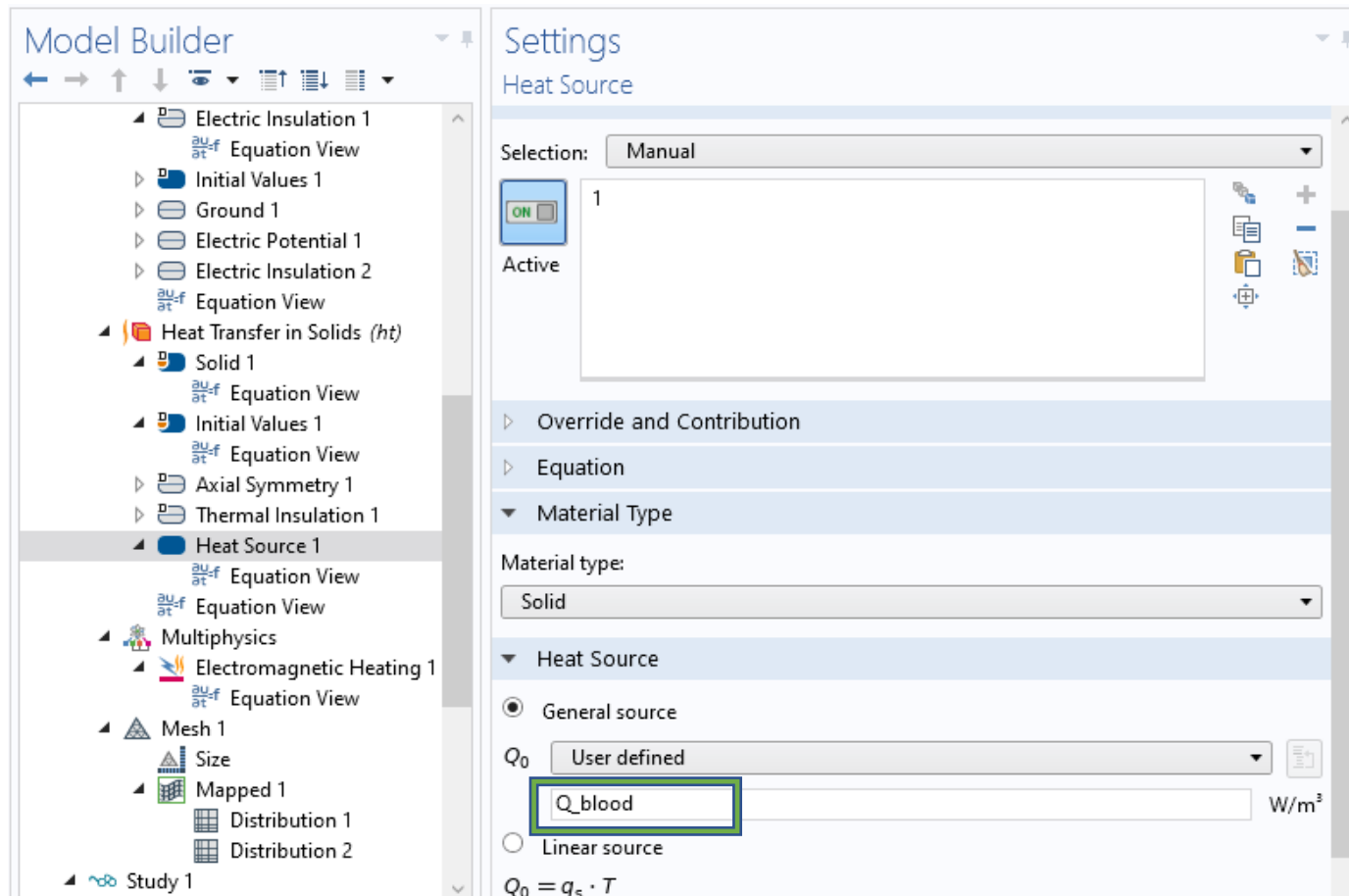
INITIAL VALUE

- 310.15 K is set as initial temperature



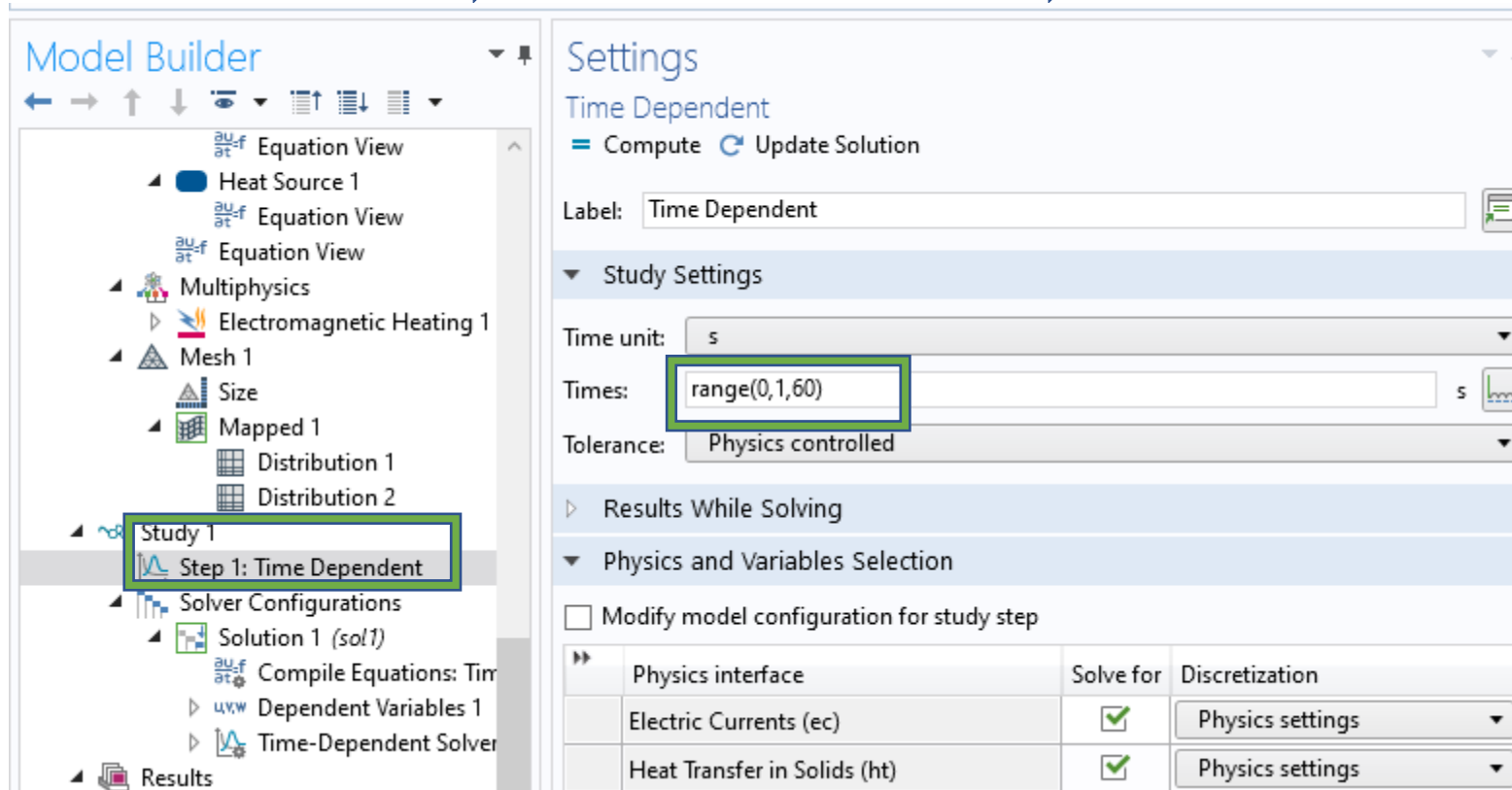
BOUNDARY CONDITION

- Heat source is defined under **Heat Transfer in solid**



STUDY

- Time Dependent  Change range  Compute




The screenshot shows the COMSOL Multiphysics interface. On the left is the **Model Builder** tree, and on the right is the **Settings** panel for the selected study.

Model Builder:

- Equation View
 - Heat Source 1
- Equation View
 - Multiphysics
 - Electromagnetic Heating 1
- Mesh 1
 - Size
 - Mapped 1
 - Distribution 1
 - Distribution 2
- Study 1 (highlighted with a green box)
 - Step 1: Time Dependent (highlighted with a green box)
 - Solver Configurations
 - Solution 1 (sol1)
 - Compile Equations: Time Dependent
 - Dependent Variables 1
 - Time-Dependent Solver
 - Results

Settings:

Time Dependent
= Compute  Update Solution

Label: Time Dependent

Study Settings

Time unit: s

Times: range(0,1,60) s (highlighted with a green box)

Tolerance: Physics controlled

Results While Solving

Physics and Variables Selection

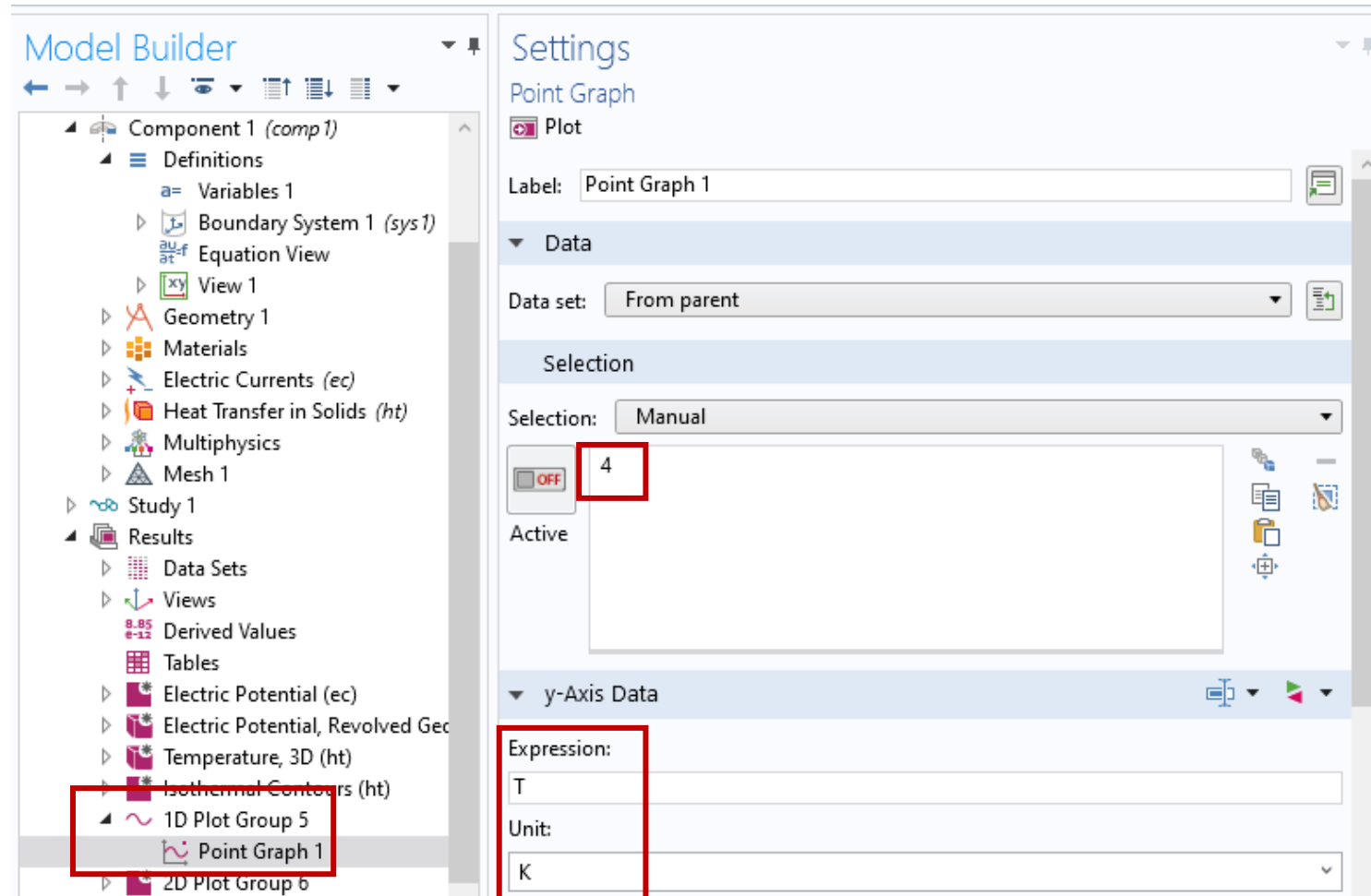
☐ Modify model configuration for study step

Physics interface	Solve for	Discretization
Electric Currents (ec)	<input checked="" type="checkbox"/>	Physics settings
Heat Transfer in Solids (ht)	<input checked="" type="checkbox"/>	Physics settings

POST PROCESSING

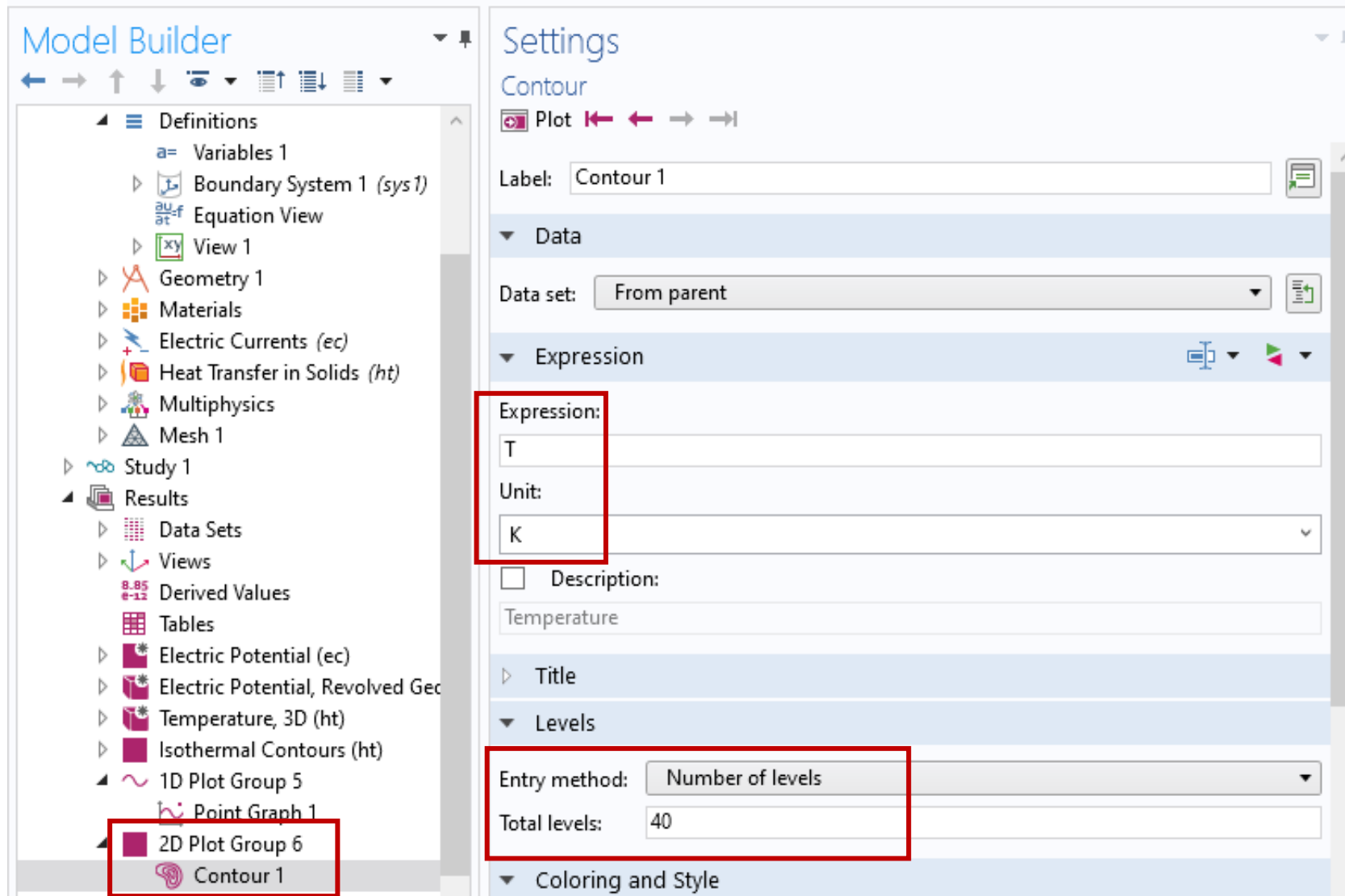
TEMPERATURE VS TIME PLOT

- 1D Plot>Domain plot
parameters>Point (4 selected)
- Y Axis – Temperature (K)
- X Axis – Time (s)



POST PROCESSING

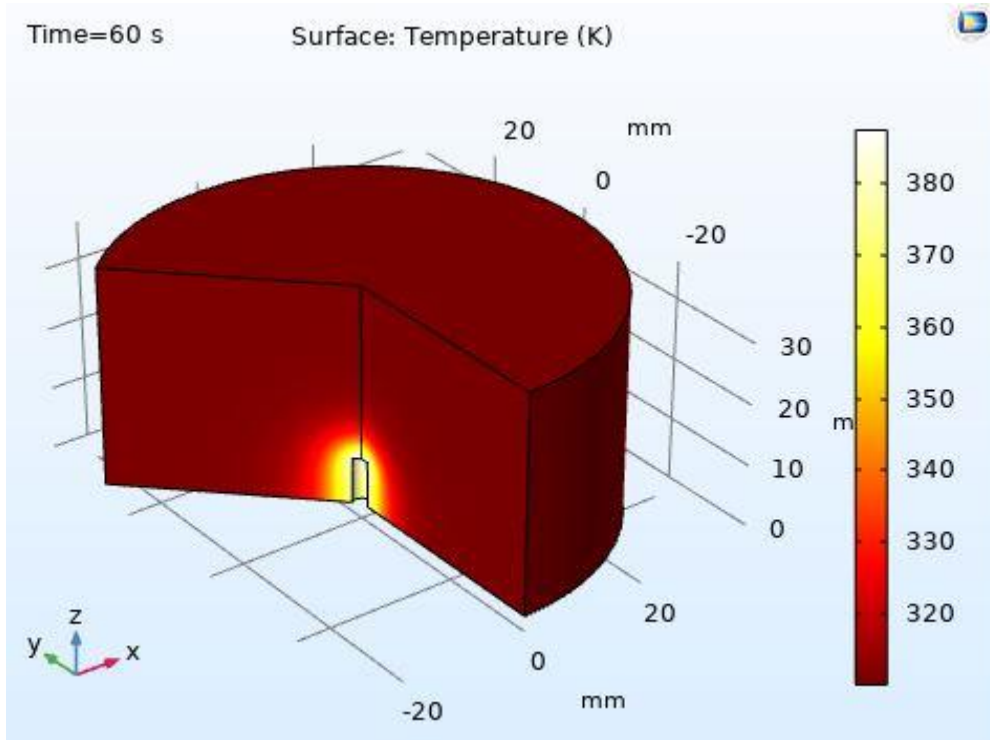
ISOTHERMAL CONTOURS



PHILOSOPHY OF THE SIMULATION

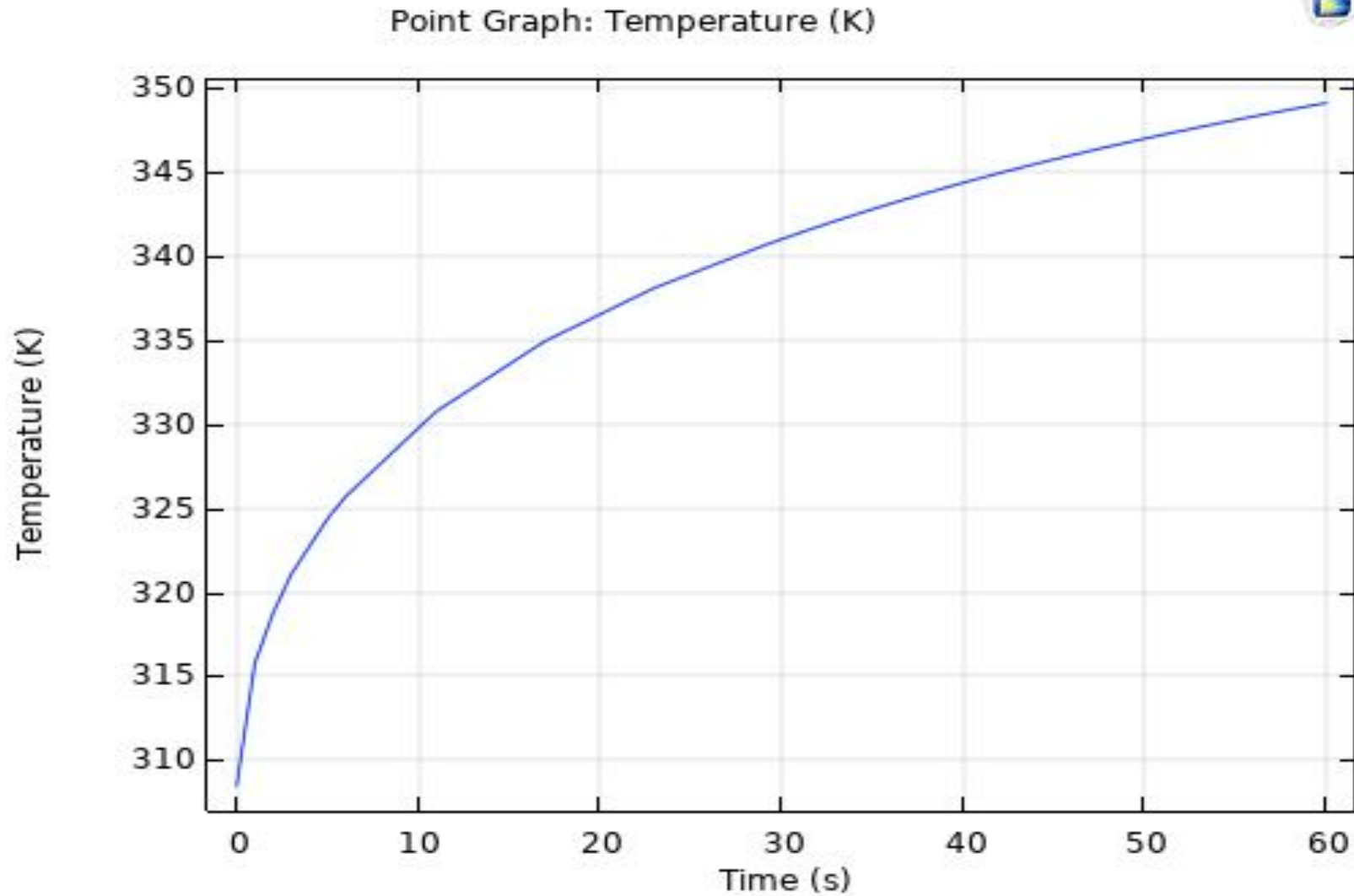
- To observe the variation of the temperature of the tissue
- Range: 37°C to more than 50°C
- Observe when the desired myocardial injury takes place
- Maintain the temperatures in the tissue below 100°C to avoid unwanted phenomena
- The temperature is controlled by Voltage regulation of the electrode

SO LET'S STUDY THE OBSERVATIONS



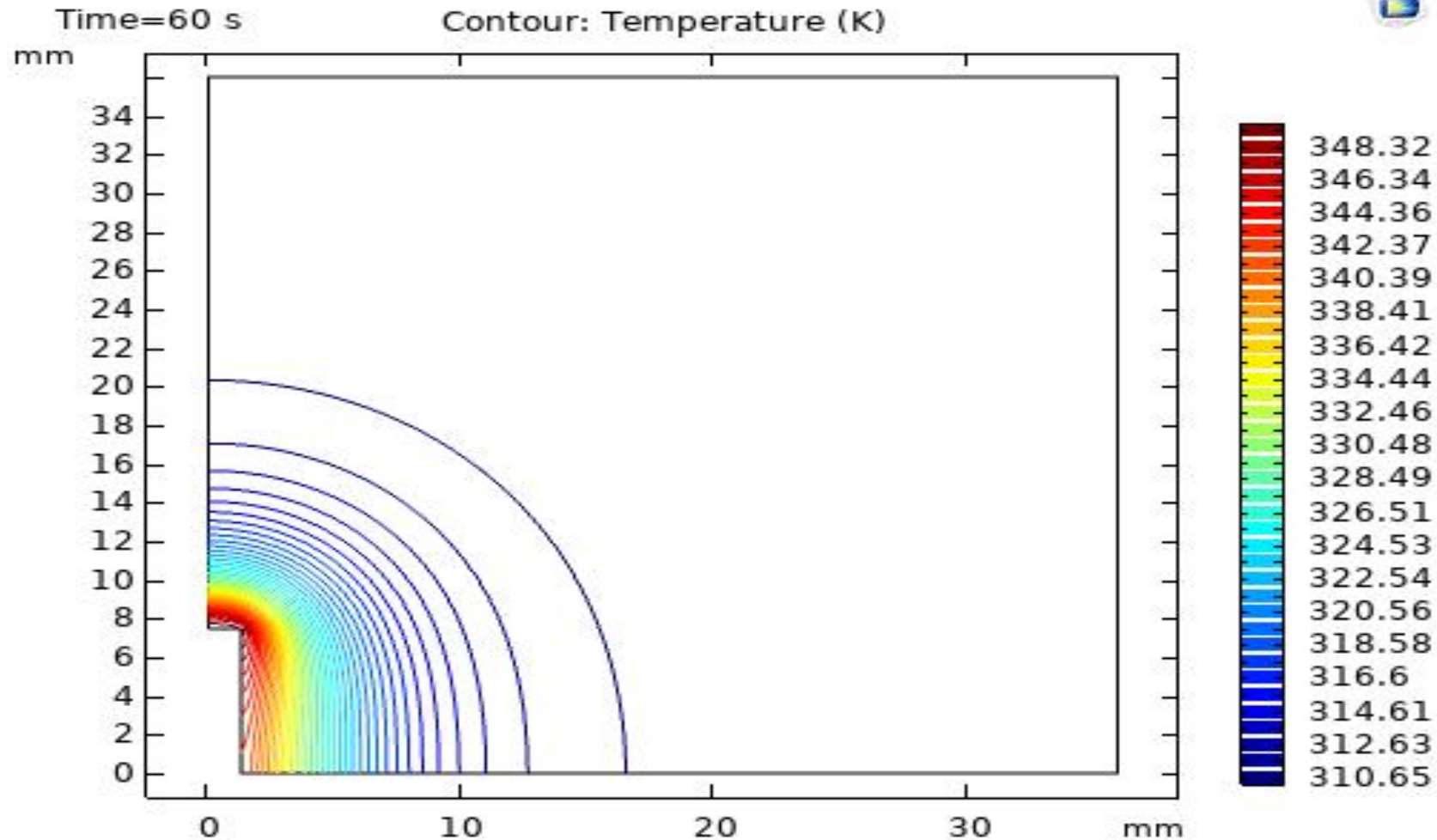
V=25V
Initial temperature =
310.15k

TEMPERATURE VS TIME GRAPH



Temperature varies from 305K to 350K.

TEMPERATURE CONTOUR



- Accurate Range
310K-348.3K
- Ablation scale
3-4 mm

ADDITIONAL SIMULATION

Objective: Adaptive Mesh Refinement

- Mesh is adapted automatically to a finer mesh while the solution runs
- Feature supported only for steady state solution

CONVERTING TO STEADY STATE SOLUTION

Heat Generation Equation by Joule Heating:

$$Q = \sigma |\nabla V|^2$$

Equivalent Heat Generation from steady state diffusion concentration:

$$Q_{dc} = 0.222 \cdot V_r \cdot V_r + 0.222 \cdot V_z \cdot V_z$$

SIMULATION STEP VARIATION

Select Physics

- AC/DC
- Acoustics
- Chemical Species Transport
 - Transport of Diluted Species (tds)**
 - Transport of Concentrated Species (tcs)
 - Chemistry (chem)
 - Reaction Engineering (re)
 - Nernst-Planck Equations (npe)
 - Nernst-Planck-Poisson Equations
 - Transport of Diluted Species in Porous Media (tds)
 - Electrophoretic Transport (el)
- Moisture Transport
- Moisture Flow
- Reacting Flow
- Reacting Flow in Porous Media

Added physics interfaces:

- Heat Transfer in Solids (ht)
- Transport of Diluted Species (tds)**

Review Physics Interface

Transport of Diluted Species (tds)

Dependent Variables

Number of species:

Concentrations:

to create new dependent variable

BOUNDARY CONDITIONS

1

The screenshot shows the Model Builder interface with the following settings for Concentration 1:

- Label:** Concentration 1
- Boundary Selection:** Manual
- Selection:** Manual
- Active:** ☒ (with a red box around the '2' and '4' in the selection list)
- Override and Contribution:** (expanded)
- Equation:** (expanded)
- Concentration:** (expanded)
- Species V:** ☒ (with a red box around the 'c_{0,v}' and '25' in the input field)

2

The screenshot shows the Model Builder interface with the following settings for Concentration 2:

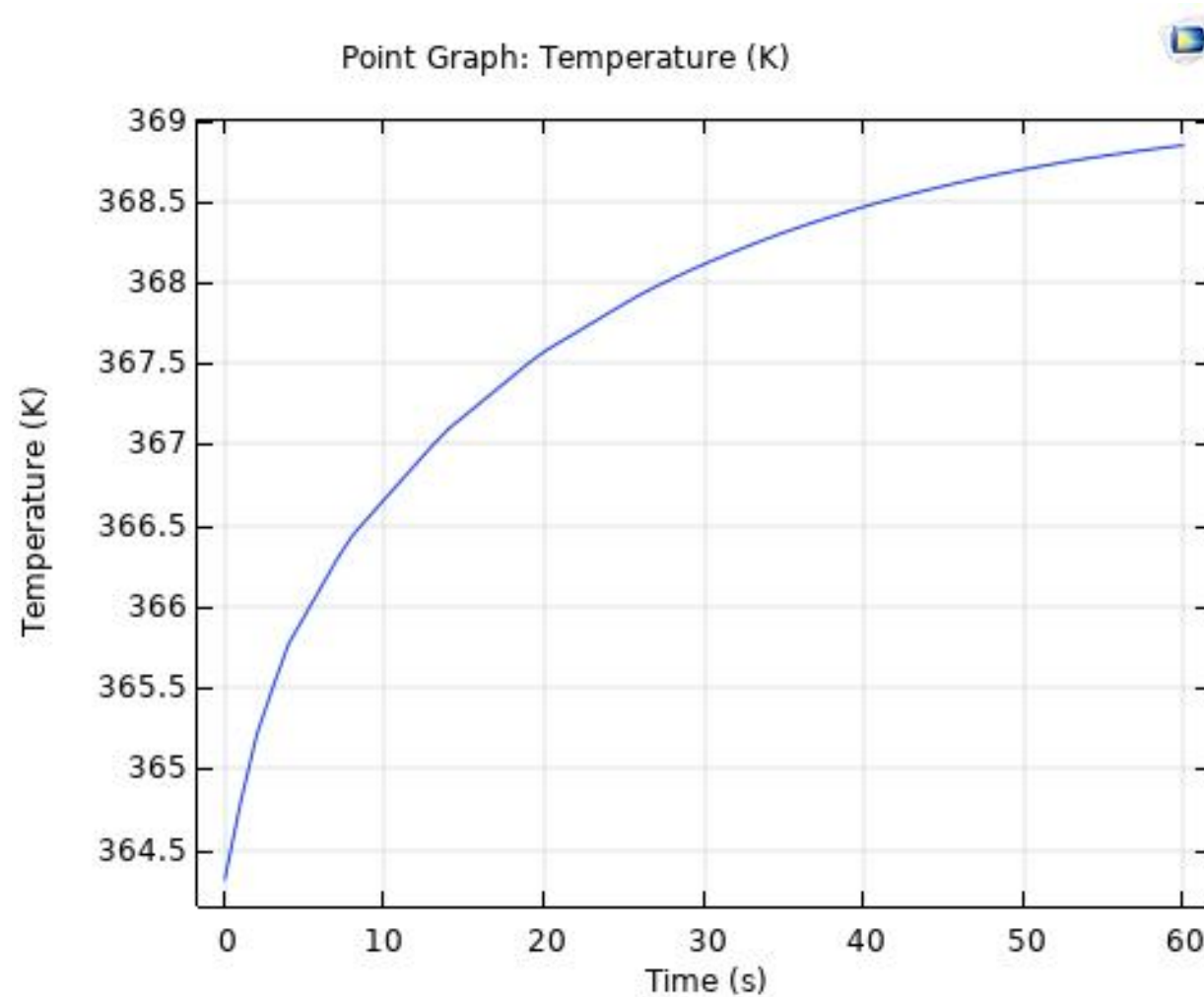
- Label:** Concentration 2
- Boundary Selection:** Manual
- Selection:** Manual
- Active:** ☒ (with a red box around the '3' and '6' in the selection list)
- Override and Contribution:** (expanded)
- Equation:** (expanded)
- Concentration:** (expanded)
- Species V:** ☒ (with a red box around the 'c_{0,v}' and '0' in the input field)

INPUT ALTERNATE EQUATION

The screenshot displays the COMSOL Multiphysics interface. On the left, the 'Component 1 (comp1)' tree shows the 'Definitions' section expanded, with 'a= Variables 1' selected. The 'Equation View' is active for the 'Transport of Diluted Species (tds)' physics. On the right, the 'Variables' table is shown, listing several variables and their corresponding expressions and units. The 'Q_dc' variable is highlighted with a red box.

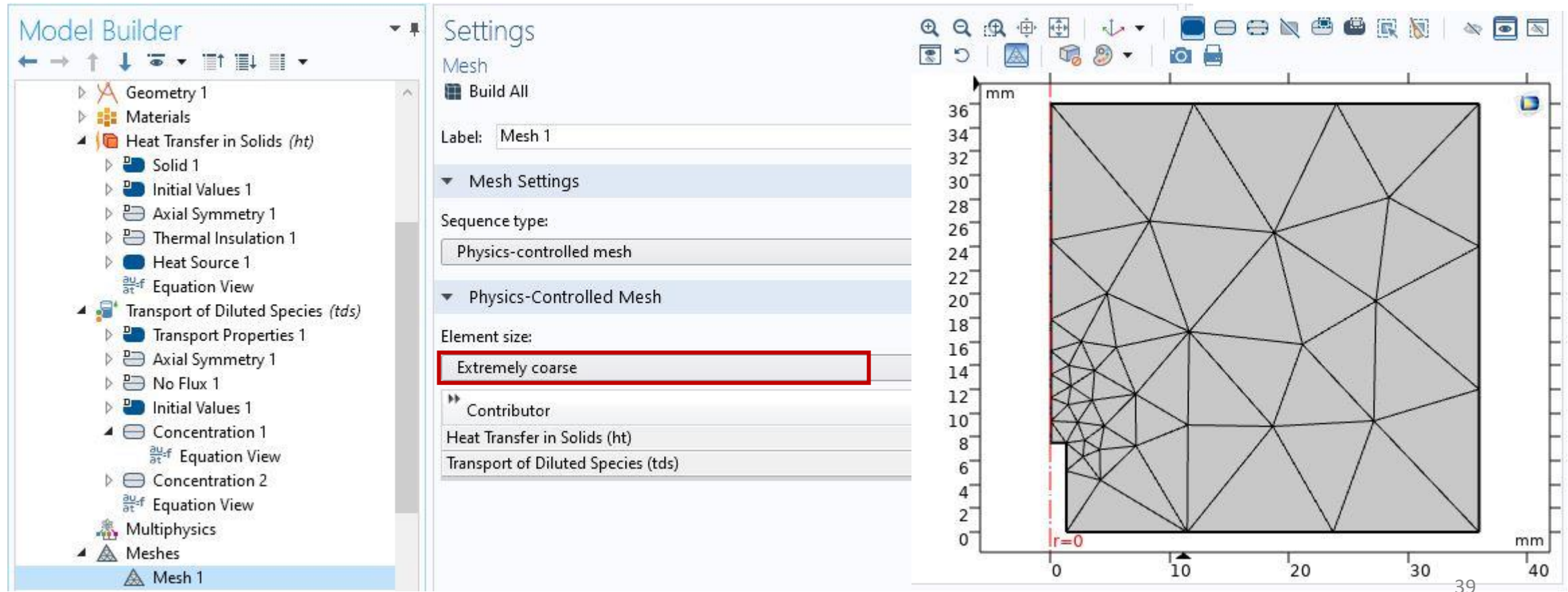
Name	Expression	Unit	Description
k	$a \cdot (0.4925 + 0.001195 \cdot T)$	W/(m·K)	
a	$1[\text{W}/(\text{K}^2 \cdot \text{m})]$	kg·m/(s...	
b	$1[\text{W}/(\text{K} \cdot \text{m}^3)]$	W/(m³·K)	
Q_blood	$b \cdot (2000 \cdot (T - 310.15))$	W/m³	
Q_dc	$c \cdot (0.222 \cdot V_r \cdot V_r + 0.222 \cdot V_z \cdot V_z)$	W/m³	
c	$1[(\text{W} \cdot \text{m}^5)/\text{mol}^2]$	kg·m⁷/(...	

TIME DEPENDENT SOLUTION



STATIONARY SOLUTION

Mesh is initially set at 'Extremely Coarse' when solution is started:



ADAPTIVE MESH SETTINGS

The screenshot displays the COMSOL Multiphysics interface, specifically the Model Builder and Settings panels. In the Model Builder, the 'Study 1' node is expanded, and 'Step 1: Stationary' is highlighted with a red rectangle. The Settings panel for 'Stationary' is open, showing various configuration options. The 'Adaptation and Error Estimates' section is expanded, and the 'Maximum number of adaptations' and 'Maximum number of elements' fields are highlighted with a red rectangle. The 'Maximum number of adaptations' is set to 3, and the 'Maximum number of elements' is set to 20000.

Model Builder

- Equation View
 - Concentration 2
 - Equation View
- Multiphysics
- Meshes
 - Mesh 1
 - Level 1 Adapted Mesh 1
 - Level 2 Adapted Mesh 2
 - Level 3 Adapted Mesh 3
- Study 1
 - Step 1: Stationary**
 - Step 2: Time Dependent
- Solver Configurations
- Results
 - Data Sets
 - Views
 - Derived Values
 - Tables
 - Temperature, 3D (ht)
 - Isothermal Contours (ht)
 - Concentration (tds)
 - Concentration (tds) 1
 - 1D Plot Group 5
 - Temperature, 3D (ht) 1
 - Isothermal Contours (ht) 1
 - Concentration (tds) 2
 - Concentration (tds) 3

Settings

Stationary

Compute Update Solution

values or dependent variables

Mesh Selection

Adaptation and Error Estimates

Adaptation and error estimates: Adaptation and error estimates

Adaptation in geometry: Geometry 1

Error estimate: L2 norm of error squared

Scaling factor: 1

Stability estimate derivative order: 2

Residual order: ☐ 0

Solution selection: Use last

☒ Save solution on every adapted mesh

Mesh adaptation

Adaptation method: General modification

☒ Allow coarsening

Element selection: Rough global minimum

Element count growth factor: 1.7

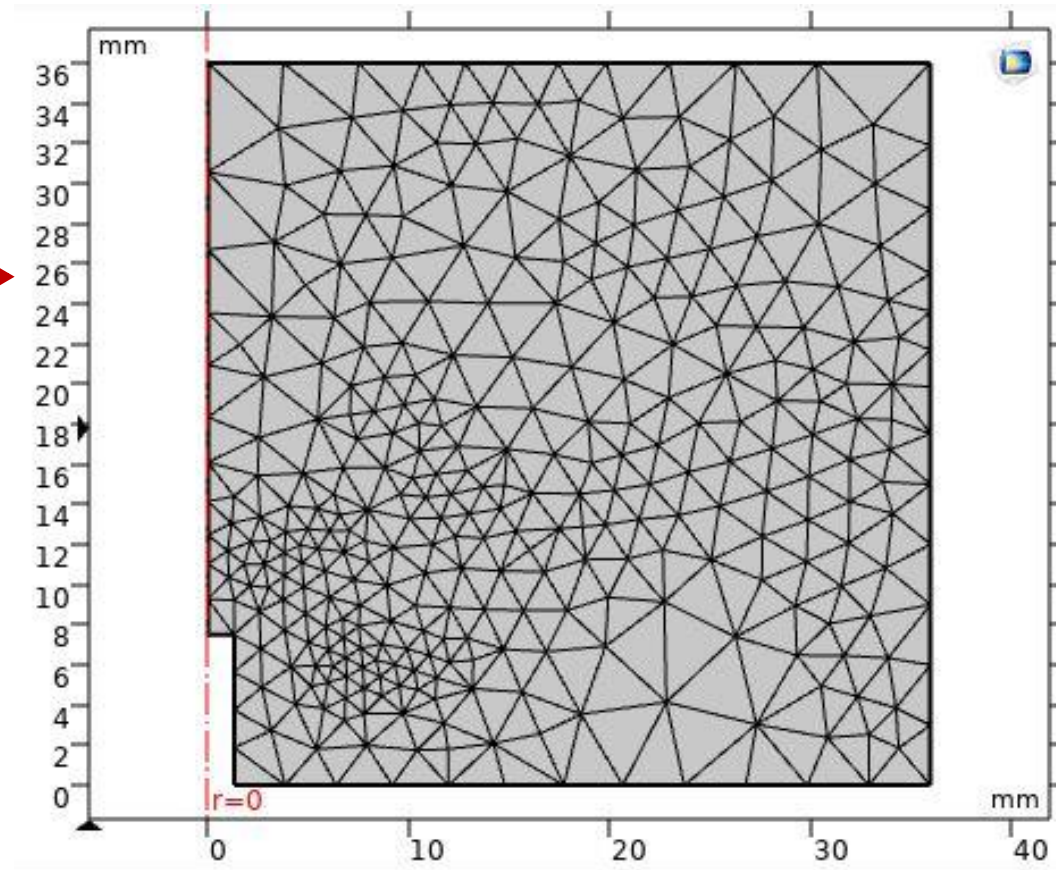
Maximum number of adaptations: 3

Maximum number of elements: 20000

Study Extensions

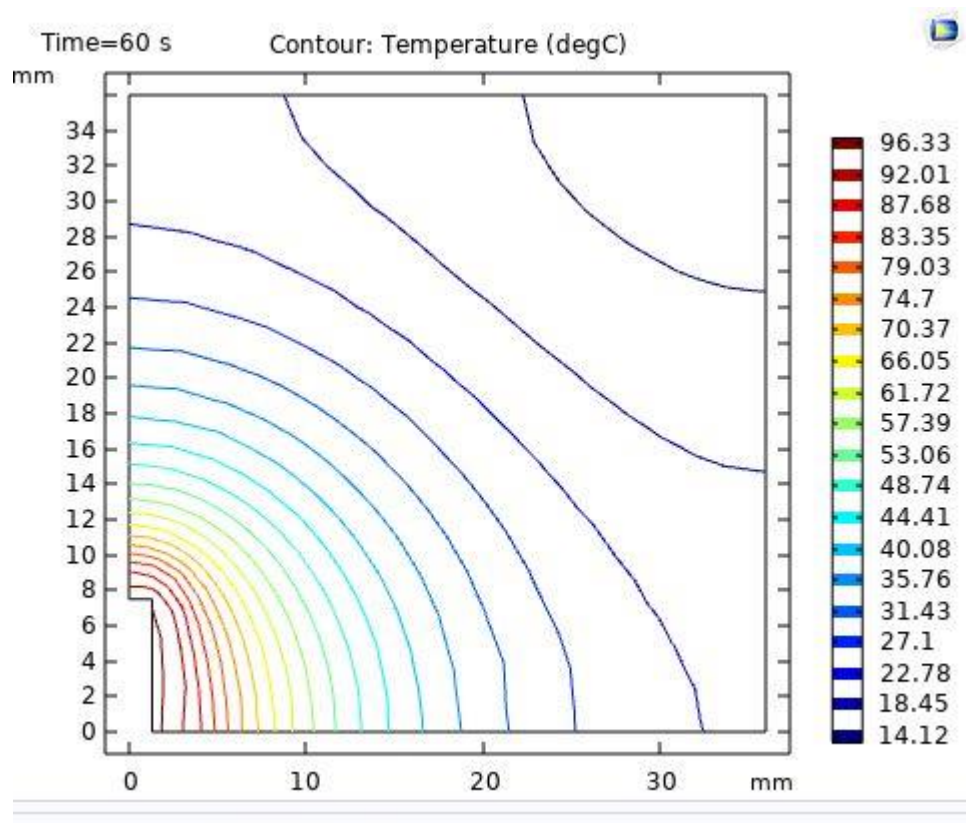
FINAL ADAPTED MESH AFTER SOLUTION

- ▲ Meshes
 - ▲ Mesh 1
 - ▲ Level 1 Adapted Mesh 1
 - ▲ Level 2 Adapted Mesh 2
 - ▲ Level 3 Adapted Mesh 3

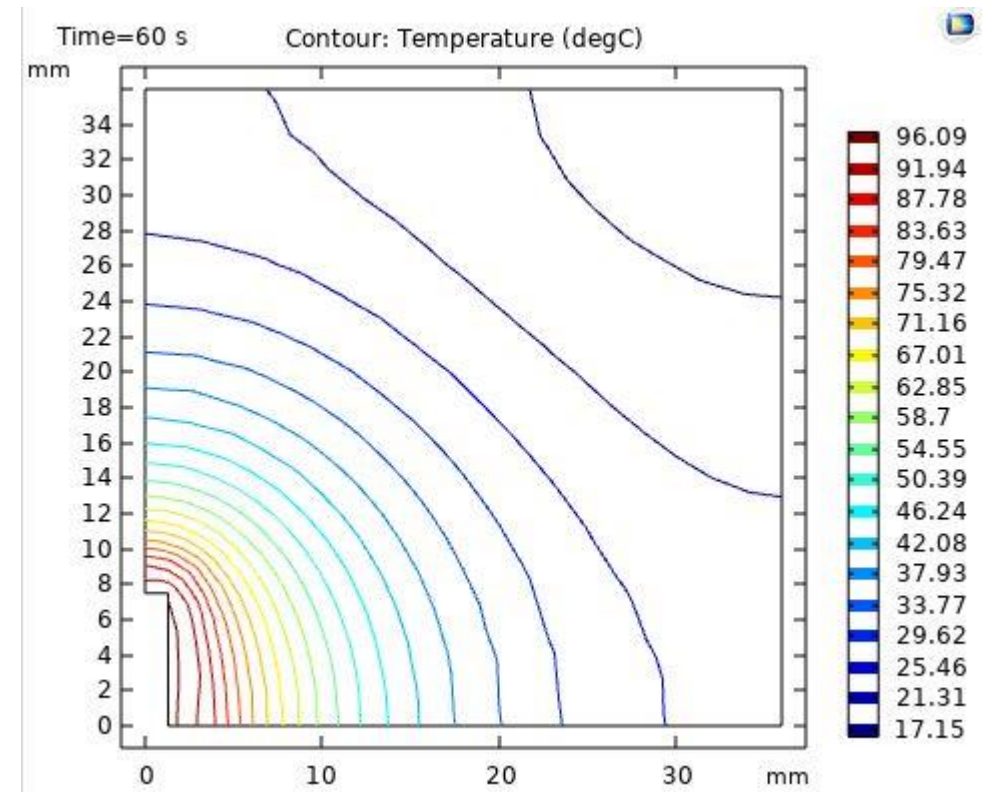


RESULT VARIATION

At extremely coarse mesh:



At final adaptive mesh:



TEAM MEMBERS AND CONTRIBUTION

1818007
Atkia Atia



Post Processing &
Result Analysis

1818017
Asfina Hassan



Simulation of
2nd Method
& Analysis

1818022
Fahmida Akhter



Theory,
Simulation of
1st Method
(Geometry)

1818024
Raisa Islam



Simulation of 1st
Method

THANK YOU!