# Case Study I

# 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:

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

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:

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

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



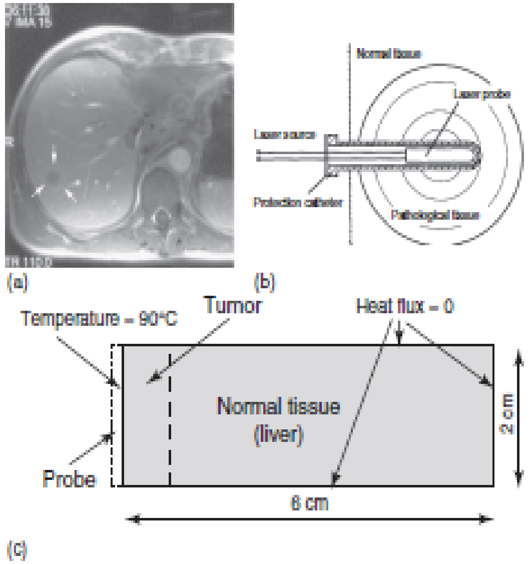



**Figure 6.1**

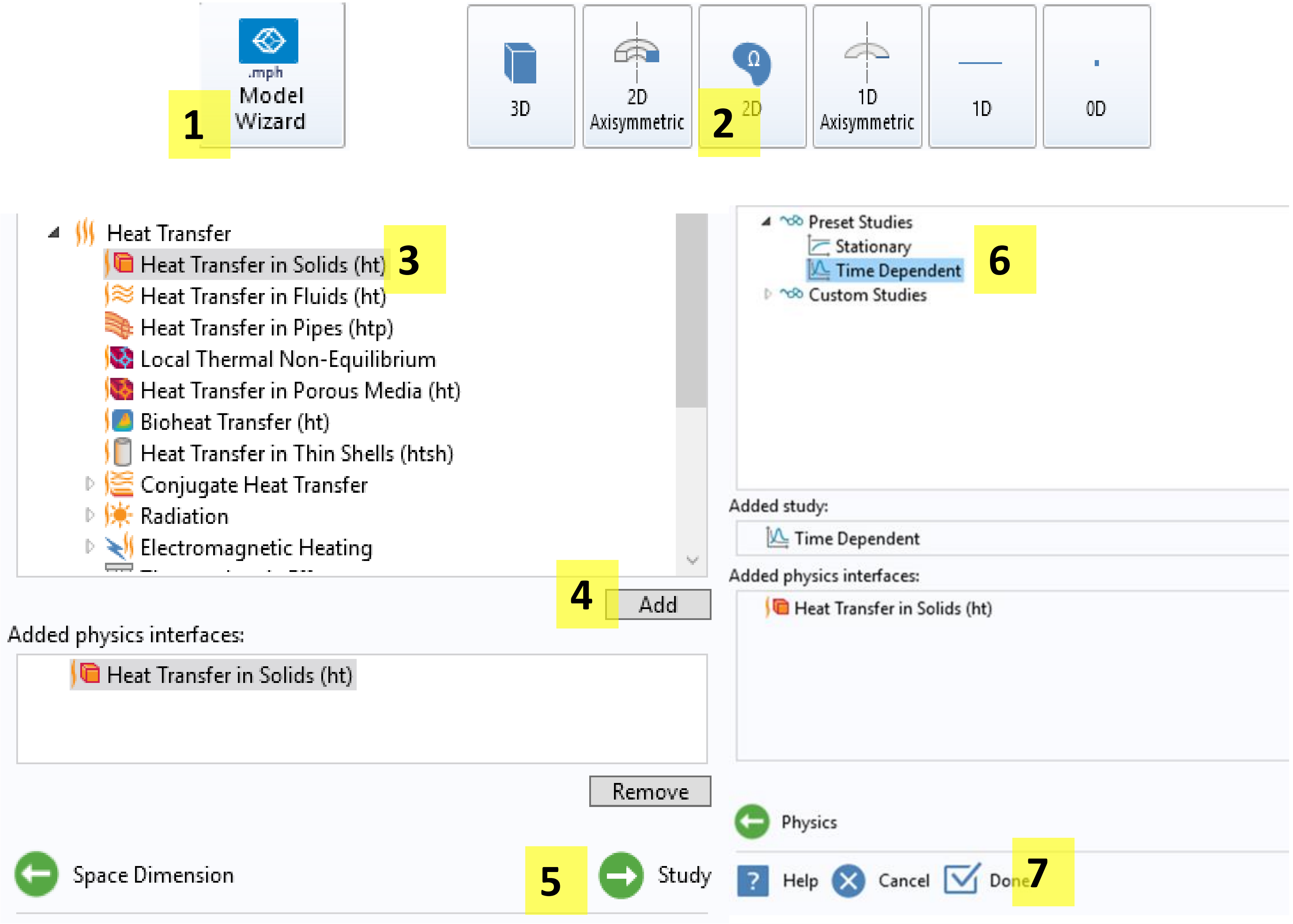
(a) A liver tumor (between the arrows); (b) schematic of the process; (c) the computational domain showing the boundary conditions. Note that the probe is not included in the computational domain. Schematic reproduced with kind permission from Springer Science+Business Media: Ma et al. (2004).

**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).

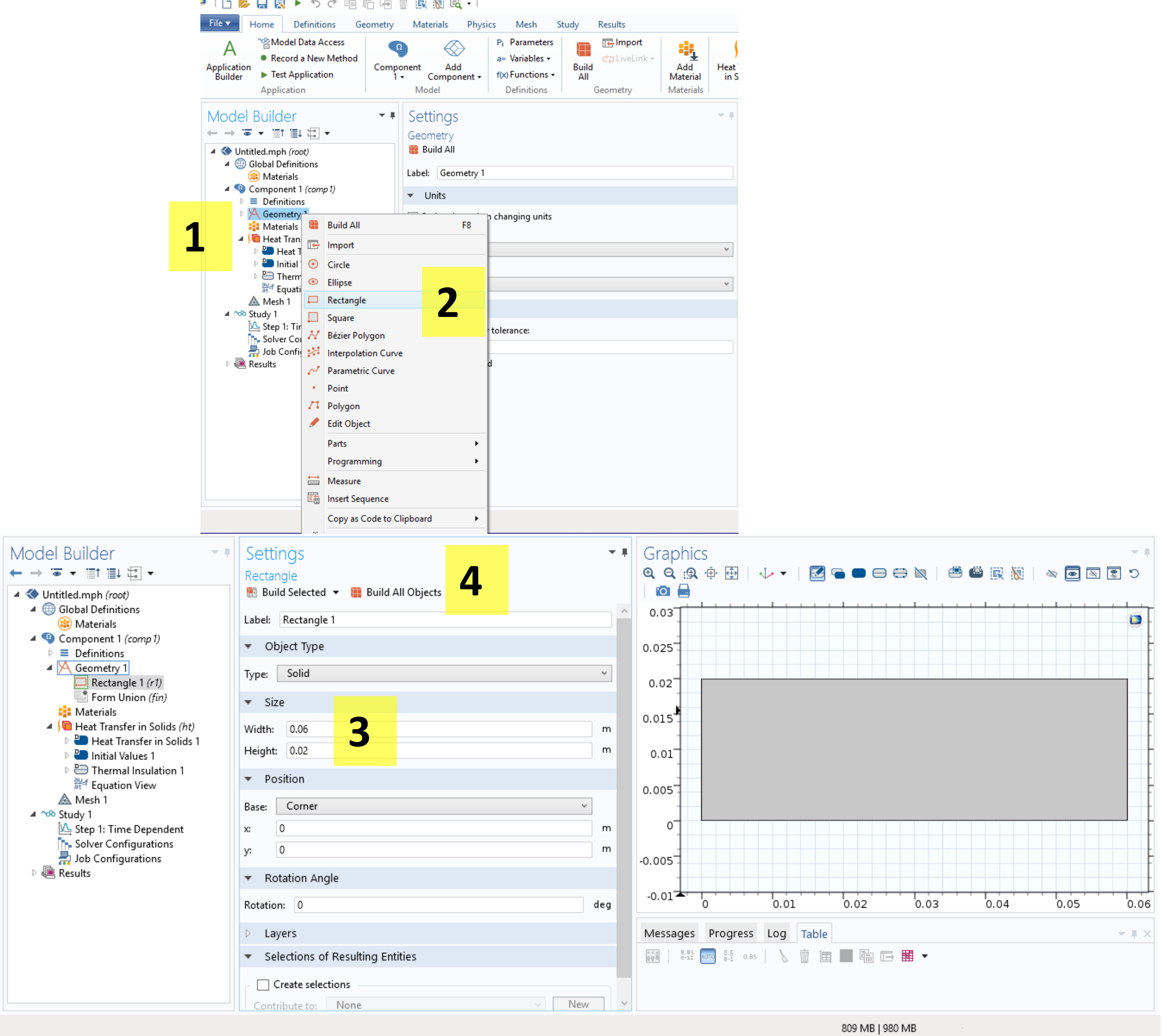
**Input parameters** The density, thermal conductivity and specific heat are 1060 kgm$^{-3}$, 0.512 W (mK)$^{-1}$ and 3600 J (kgK)$^{-1}$, respectively. The properties of the normal tissue and tumor are assumed to be the same.

## Reference

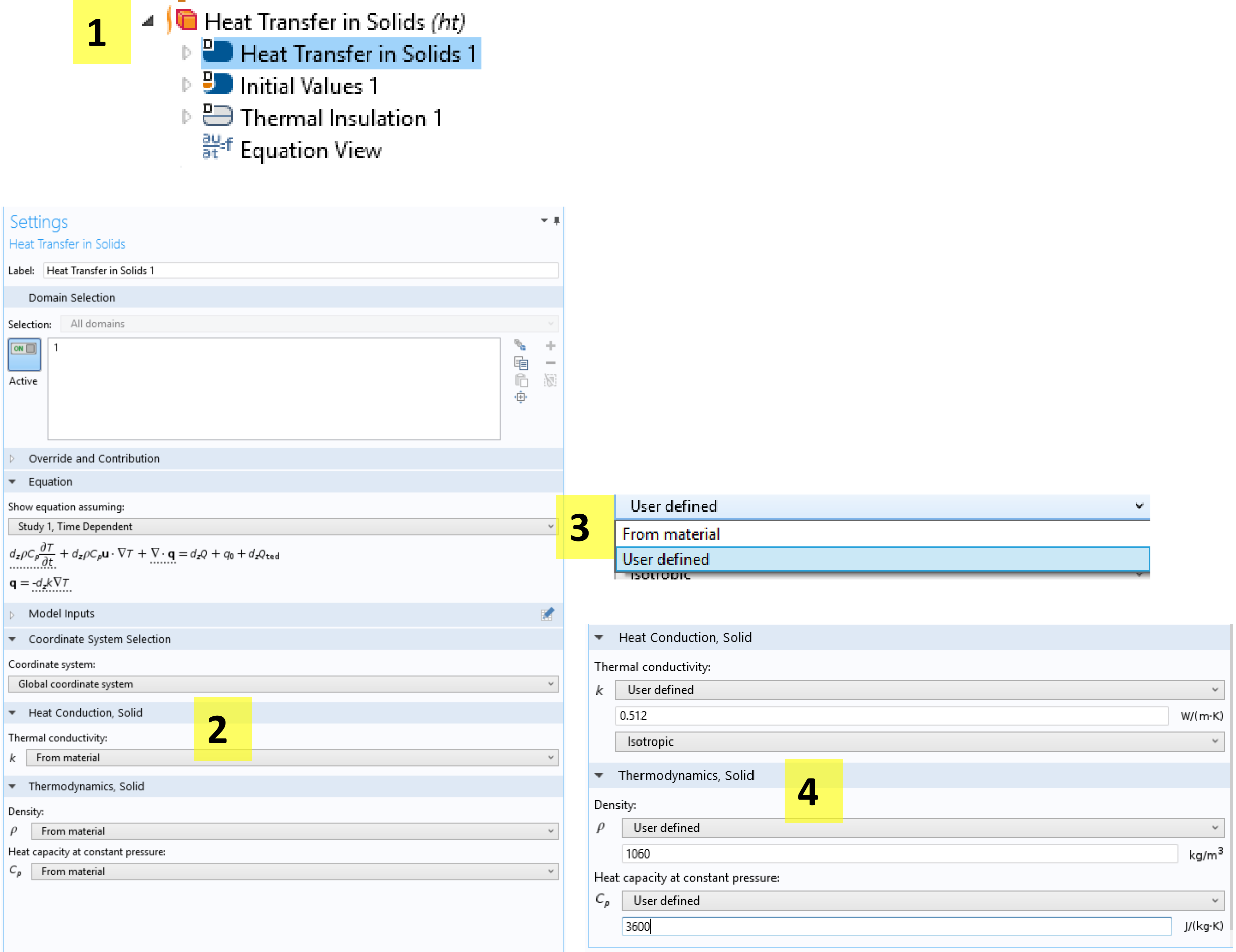
Ma, N., X. Gao, and X. X. Zhang. (2004). Two-layer simulation model of laser-induced interstitial thermo-therapy. *Lasers in Medical Science*, **18**(4): 184–189.



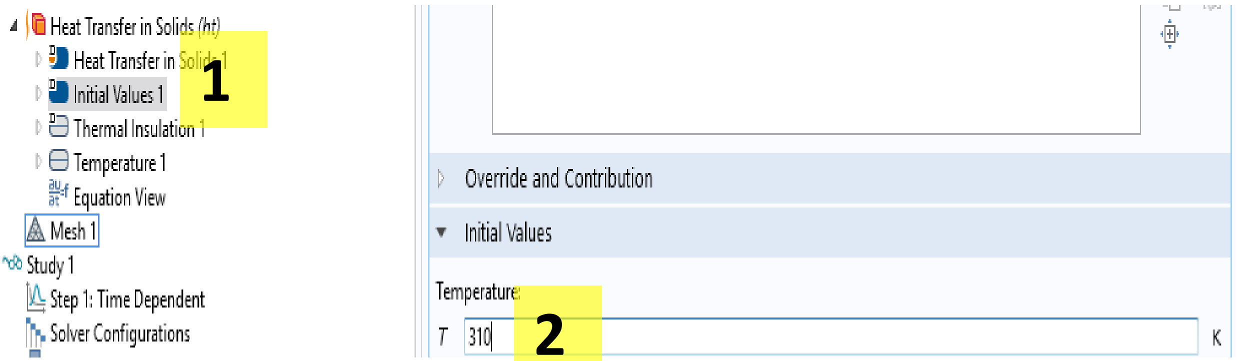

1) Select model wizard
2) Select 2D
3) Select 'Heat transfer in solids'
4) Click 'Add'
5) Click 'Study'
6) Click 'Time Dependent'
7) Click 'Done'

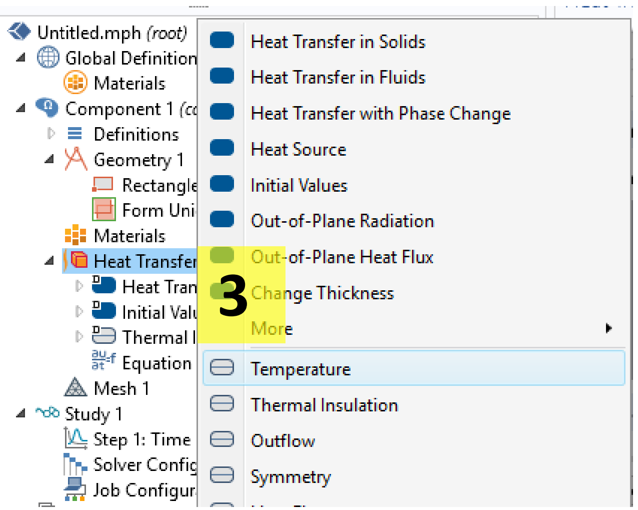


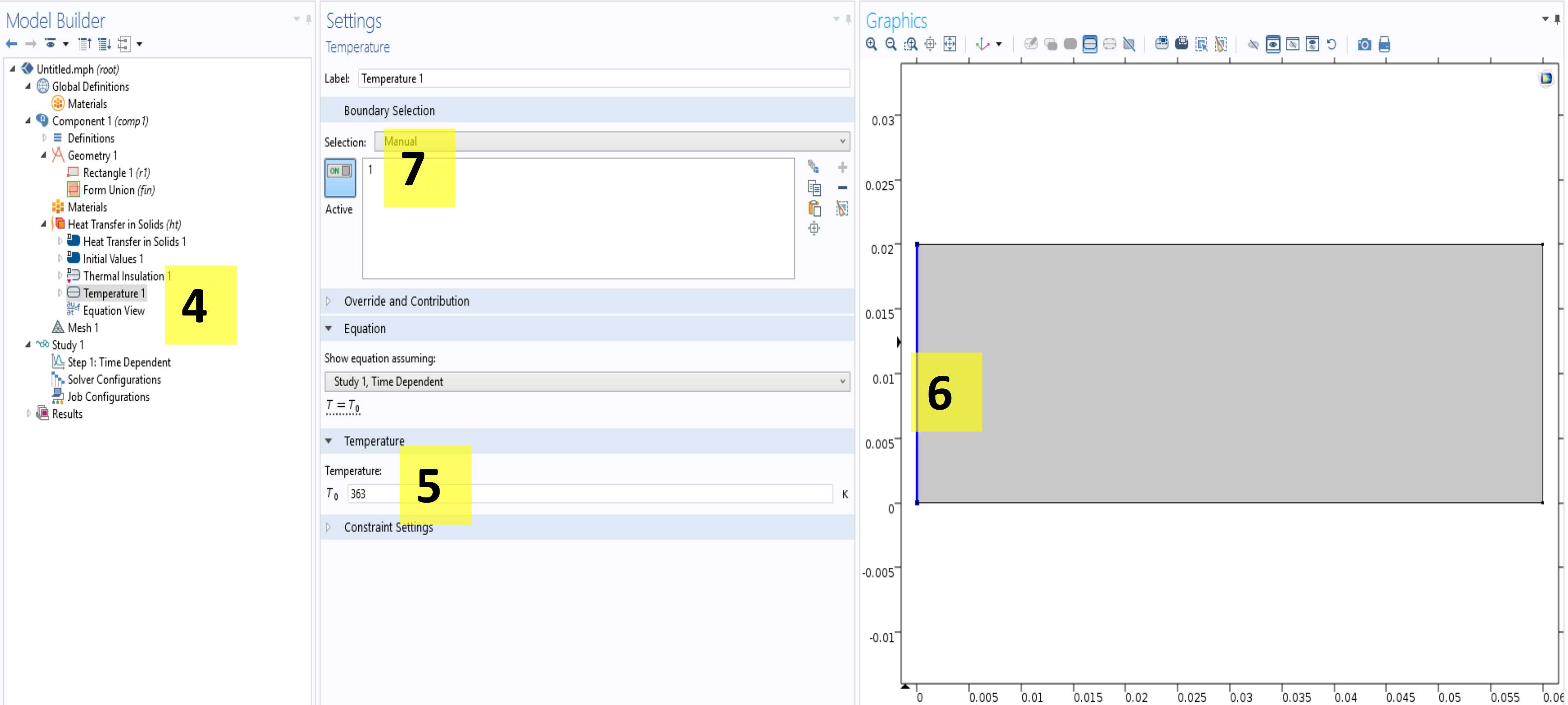
1) Right click geometry
2) Select rectangle
3) Enter width (0.06m) and height (0.02m) as shown
4) Click 'Build all objects'



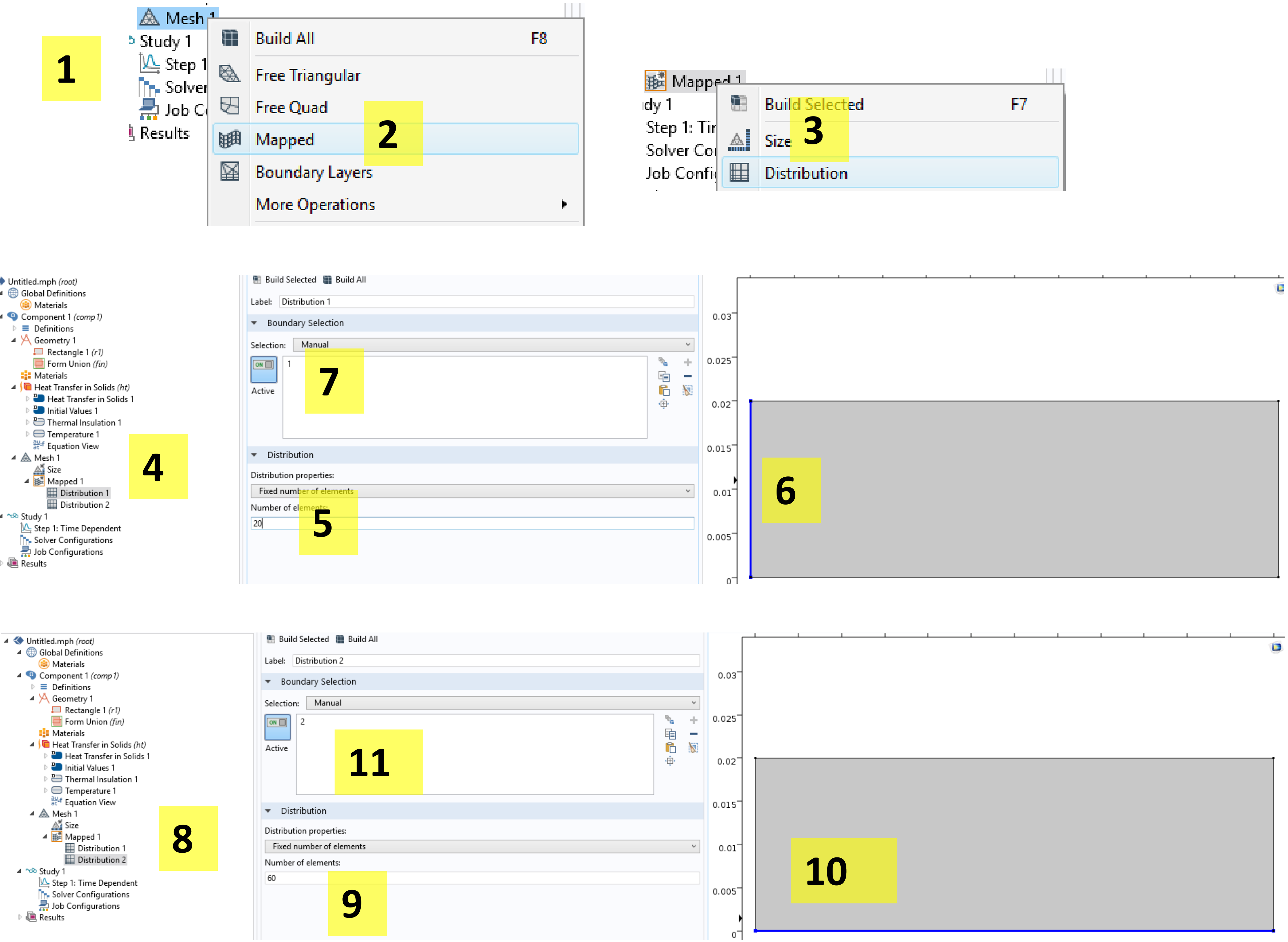

1) Go to ‘Heat transfer in solids’ >> ‘Heat transfer in solids 1’
2) In the middle panel, scroll down to k, ρ, and Cp
3) Select ‘user defined’
4) Input 0.512 W/(mK) for thermal conductivity. Input 1060 kg/m^3 for density. Input 3600 J/(kgK) for heat capacity.



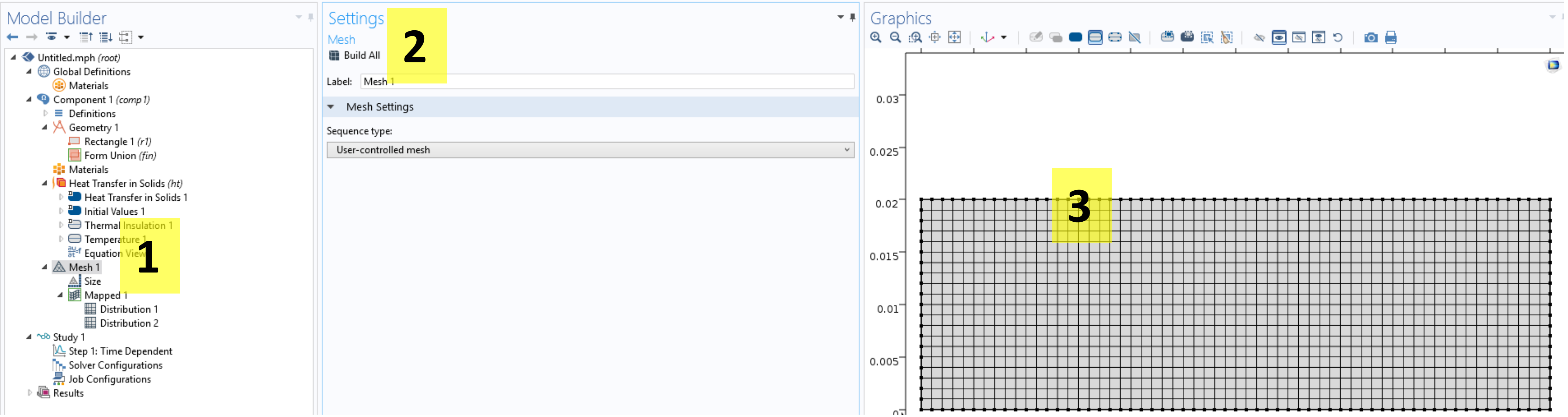




1) Left click ‘Initial values 1’
2) Enter 310 K
3) Right click ‘Heat transfer in solids’ and select ‘Temperature’
4) Left click on ‘Temperature’
5) Enter boundary temperature, 363 K
6) Left click the left boundary
7) The number ‘1’ should appear

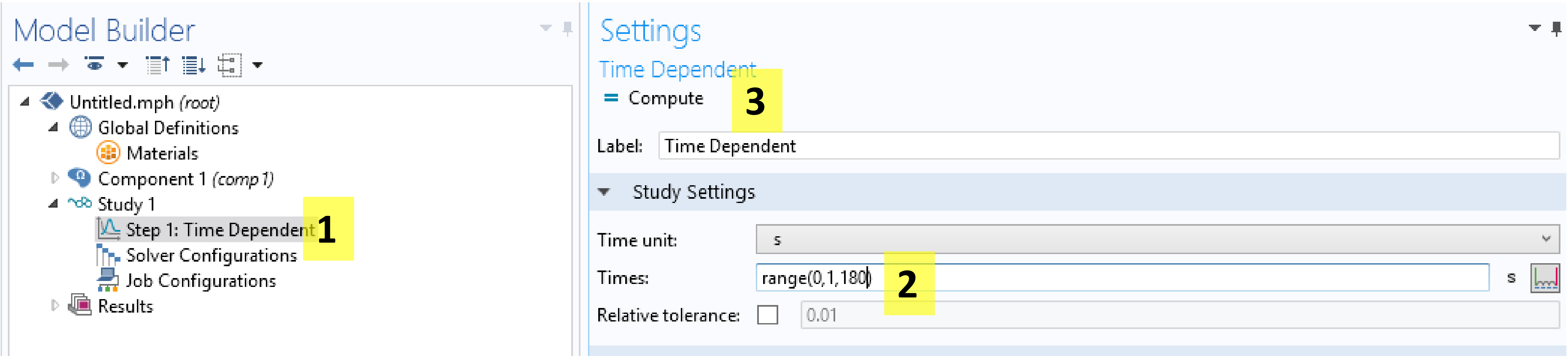


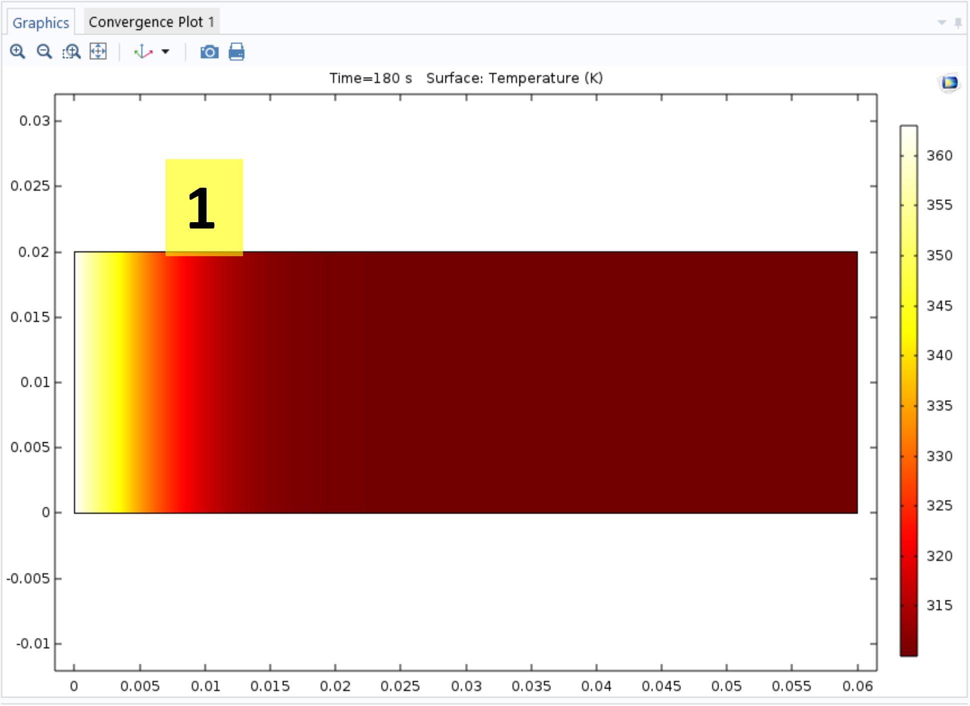

1) Right Click mesh 1
2) Click Mapped
3) Right click Mapped 1 and select 'Distribution'.
   <u>Do step 3 again</u>
4) Left Click distribution 1
5) Enter 20
6) Left click left boundary
7) You should see a 1
8) Left Click distribution 2
9) Enter 60
10) Left click bottom boundary
11) You should see a 2

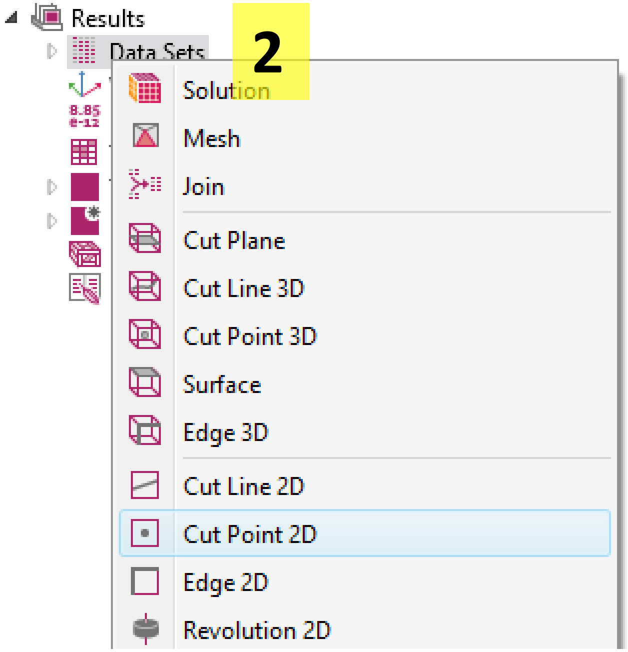



1) Click Mesh 1
2) Click Build All
3) You should see the following mesh

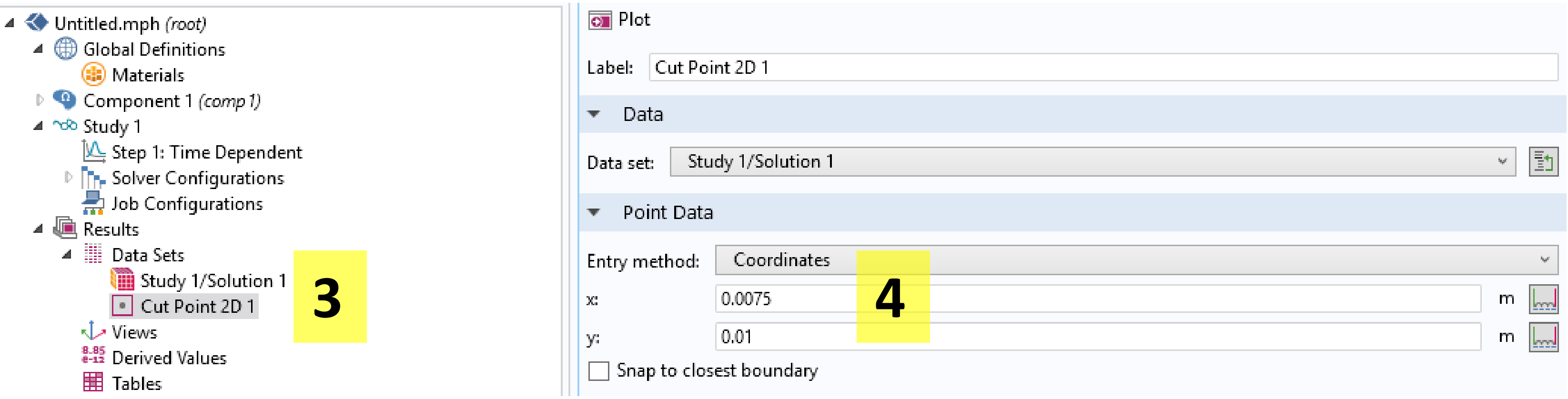
1) Left click “Step 1: Time dependent’
2) Enter range(0,1,180)
3) Click compute

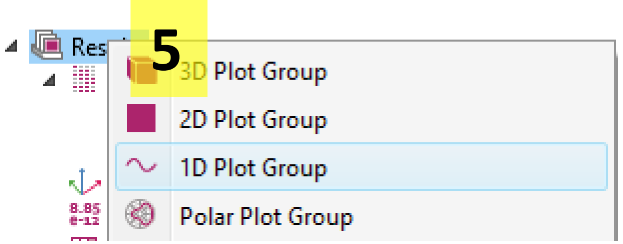


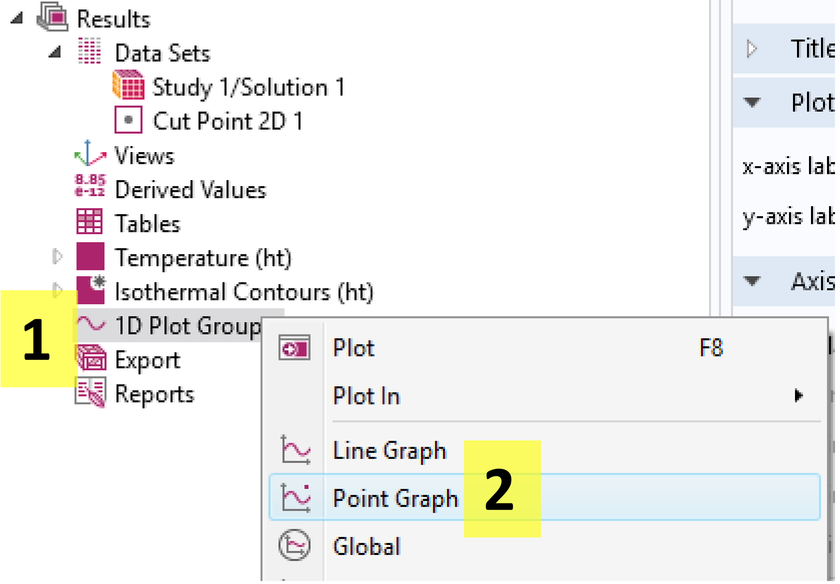


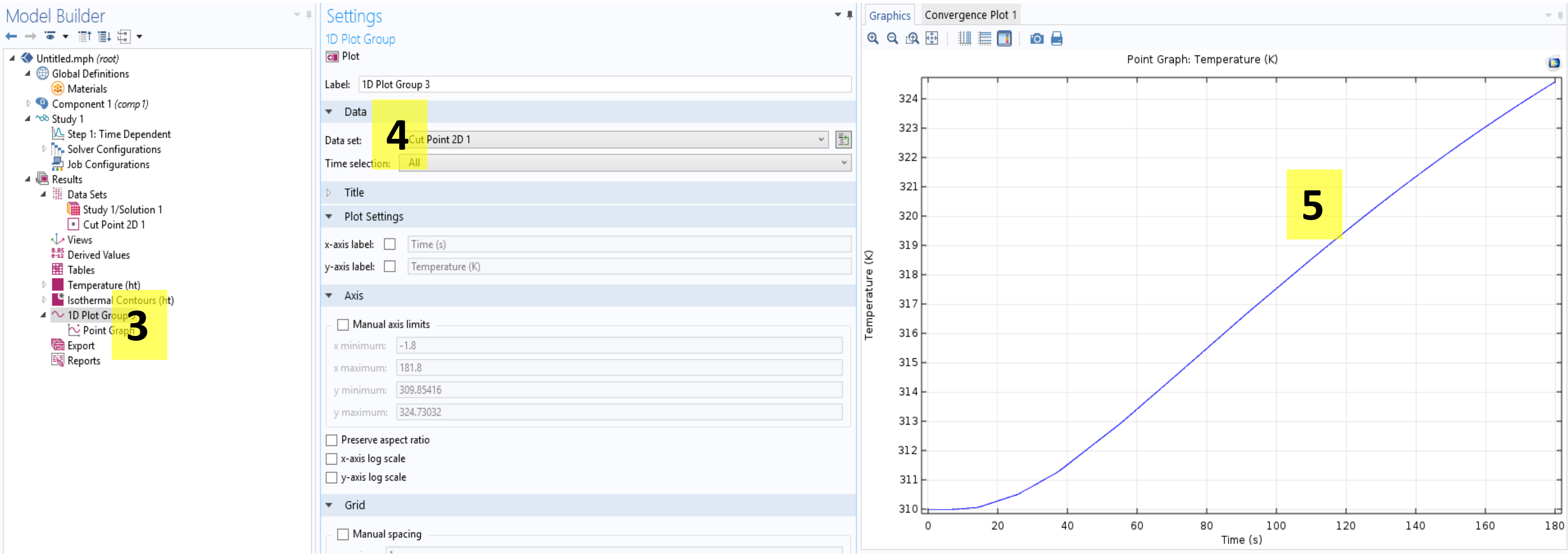

1) You should see the following surface plot
2) Right click ‘Data sets’ and select ‘Cut point 2D”
3) Left click ‘Cut point 2D 1’
4) Enter x = 0.0075 and y= 0.01
5) Right click ‘Results’ and select ‘1D plot group’



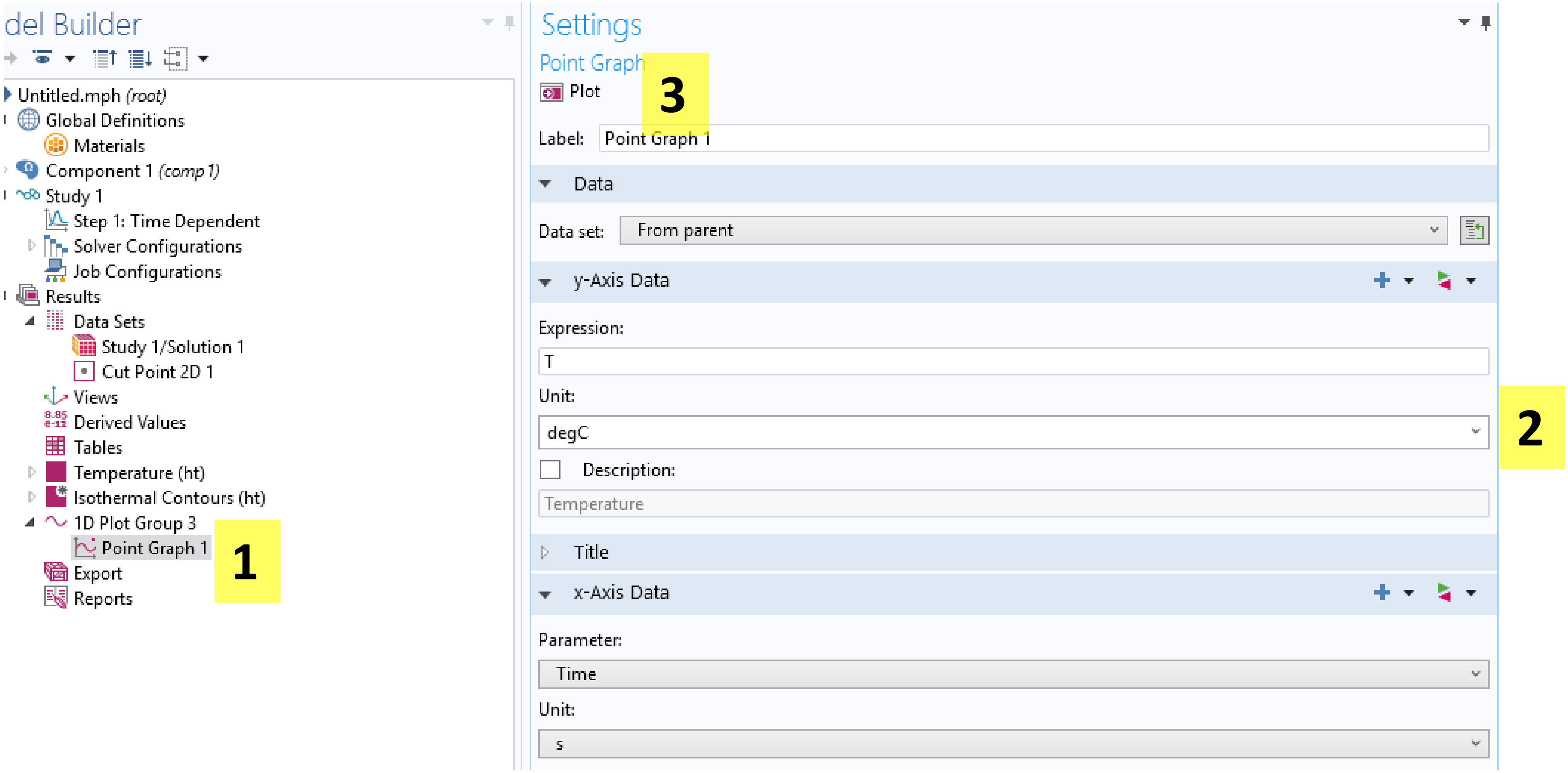






1) Right click ‘1D plot group 3’
2) Select ‘Point graph’
3) Left click ‘1D plot group 3’
4) Select ‘Cut point 2D 1’ from dropdown for ‘data set’
5) You should see this plot



1) To change units, left click ‘Point graph 1’
2) Select degC from dropdown
3) Click plot



# Case Study II

# II Cryosurgery of a wart

This case study analyzes the process of cryogenic wart treatment by optimizing the temperature and the duration of liquid jet that is applied to the surface of common warts (Cuneo *et al.*, 2002). The goal is to destroy as much of the wart as possible while damaging as little healthy skin as possible.

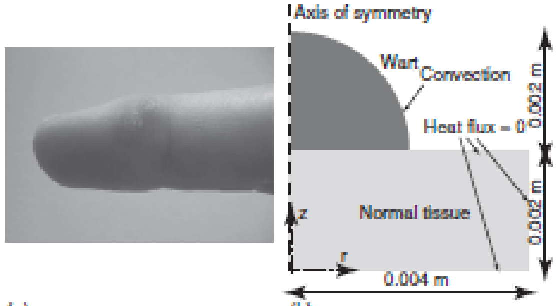
## Problem formulation

The wart as pictured in Figure 6.2(a) is simplified as a hemispherical protrusion from a flat skin surface. We assume homogenous properties and perfectly symmetrical geometry of the wart and skin. Since we want to destroy only the wart and very little of the surrounding tissue, if any, we can restrict our computational region in the normal tissue to be a symmetric region around the wart. The geometry can then be considered as axisymmetric with the axis as shown in Figure 6.2(b) and therefore, can be modeled in two dimensions so that the wart becomes a quarter-circle attached to a flat skin slab. The geometry can be swept 360 degrees around the axis to obtain the three-dimensional representation (Figure 6.2(a)).

**Governing equations** The problem involves heat transfer only and the mode of heat transfer is conduction. There is no heat transfer by convection inside the normal tissue and wart system and no heat source term. Therefore the governing equation that needs to be solved is

$$\frac{\partial T}{\partial t} = \frac{1}{\rho C_p}\left[\frac{1}{r}\frac{\partial}{\partial r}\left(kr\frac{\partial T}{\partial r}\right) + \frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)\right] \tag{6.3}$$

Notice here that since the problem is axisymmetric, the partial space derivative terms are written with respect to variables $r$ and $z$ instead of $x$ and $y$.




(a) (b)

**Figure 6.2**

(a) Two common warts on the surface of the skin on a finger; (b) the computational domain showing subdomains (wart, normal tissue) and boundary conditions. Wart figure from http://en.wikipedia.org/wiki/Image:Wart_ASA_animated.gif

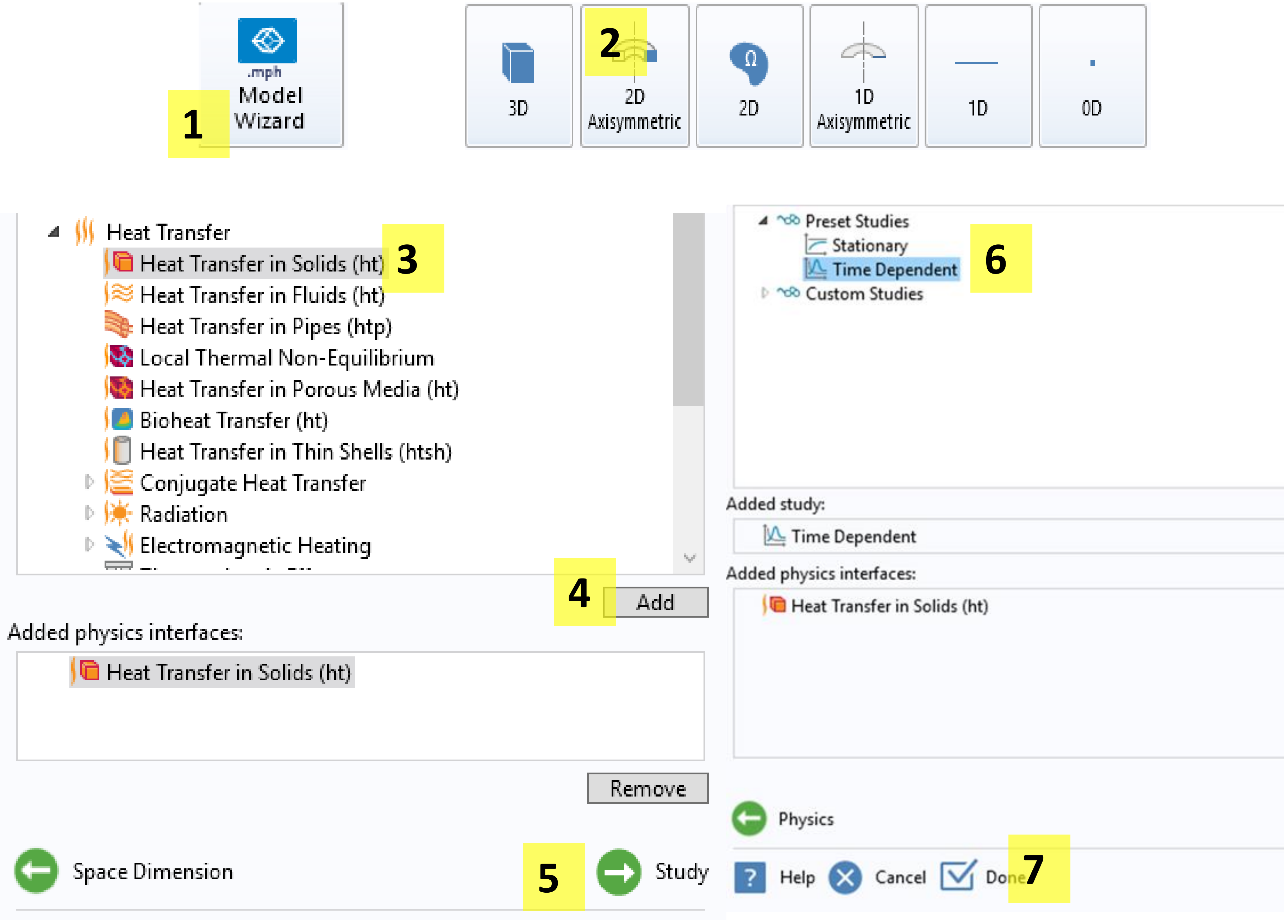


**Boundary conditions** The liquid jet is applied to the surface of the wart at $-196\,^\circ$C with a heat transfer coefficient of 5000 W m$^{-2}$K$^{-1}$. The application of the liquid jet is included as a convective boundary condition, as shown in Figure 6.2(b). The top surface of the normal tissue adjacent to the wart is insulated to the surrounding air. Additionally, the bottom and right edges of the normal tissue are assumed to be at a considerable distance so that there are no variations in temperature in those areas. Hence, a zero heat flux boundary condition is applied on the right and bottom edges (Figure 6.2(b)).

**Input parameters** Input parameters needed for the model, which include thermal conductivity and specific heat capacity, are shown in Figure 6.3. The rapid change in thermal properties in the temperature range 0 to $-5\,^\circ$C (as seen in Figure 6.3) is due to freezing of water present in the tissues. The densities of the normal tissue and wart were 1000 kgm$^{-3}$ and 1500 kgm$^{-3}$, respectively.

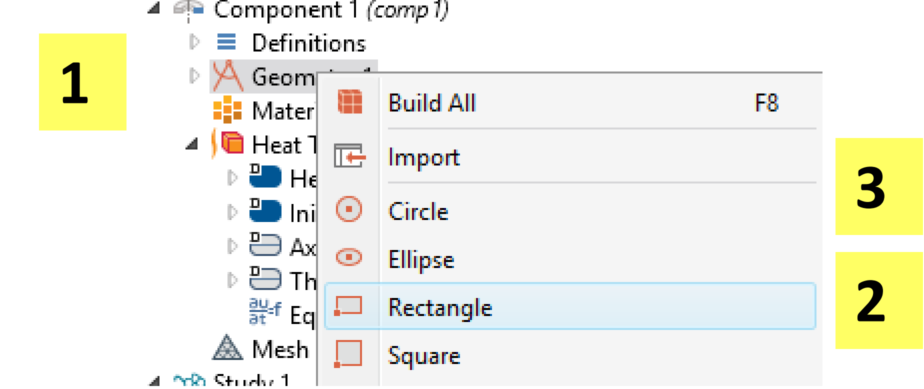
## Reference

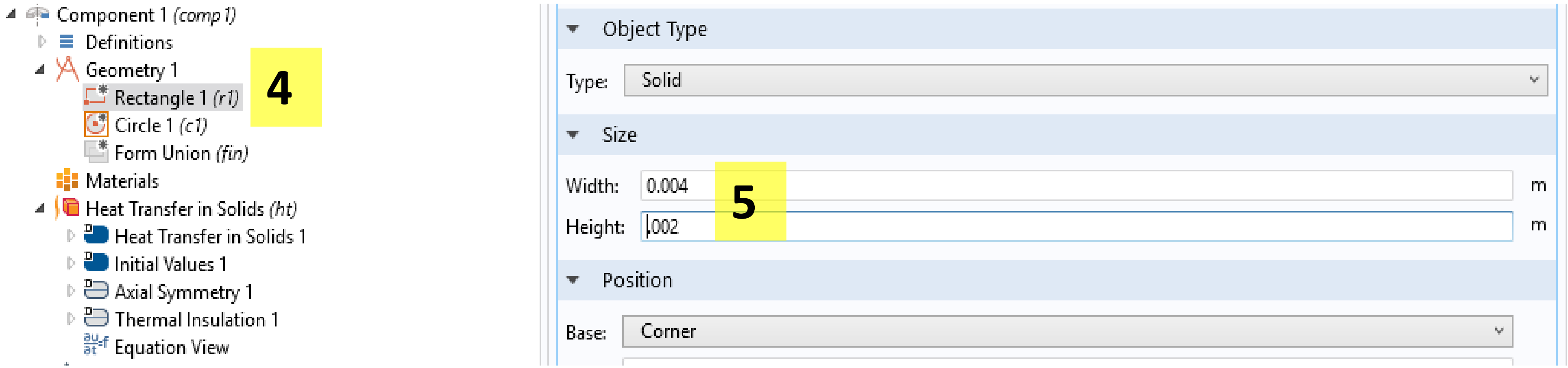
Cuneo, K., LeBarron, J., Reynolds, J., Tiberio, C. and Yoo, S. (2002). Cryogenic Treatment of the Common Wart On the web at http://ecommons.library.cornell.edu/handle/1813/212

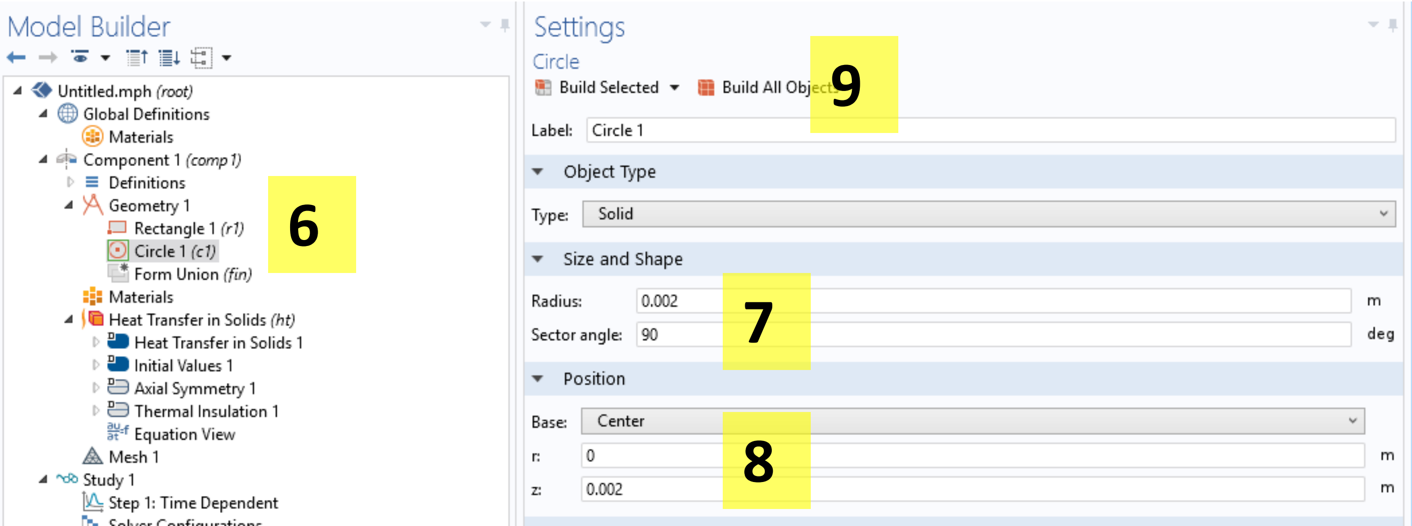


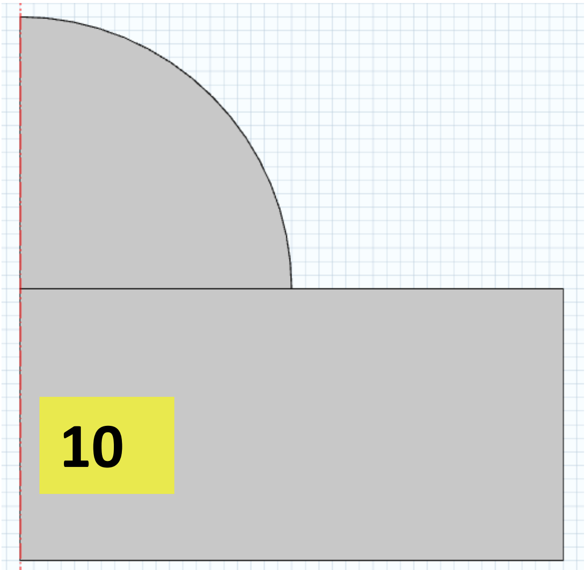

1) Select model wizard
2) Select 2D Axisymmetric
3) Select ‘Heat transfer in solids’
4) Click ‘Add’
5) Click ‘Study’
6) Click ‘Time Dependent’
7) Click ‘Done’

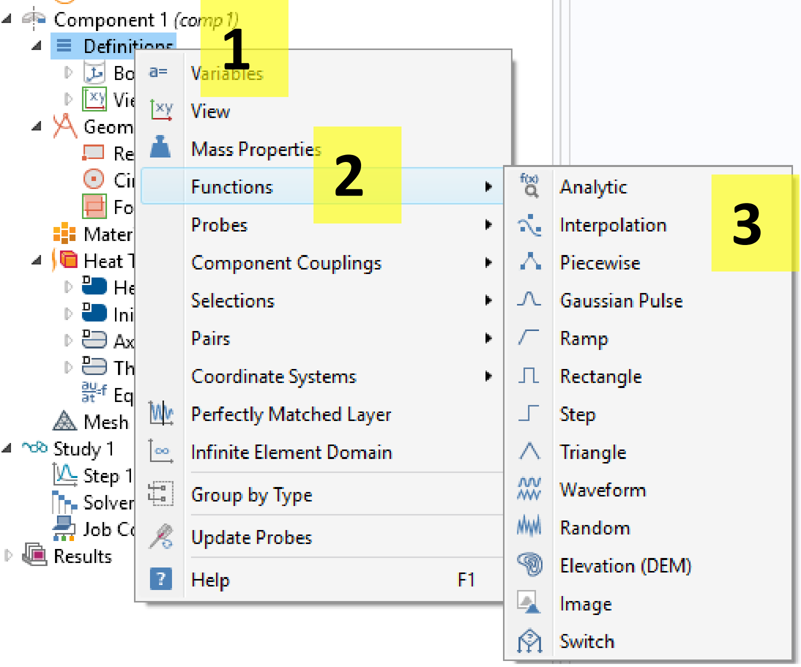
1) Right click geometry
2) Select rectangle
3) Right click geometry again and select circle
4) Left click ‘rectangle 1’
5) Enter width 0.004m and height 0.002 m
6) Left click circle
7) Enter radius 0.002, sector angle 90
8) Enter z = 0.002 for position
9) Click Build all
10) You should see this plot



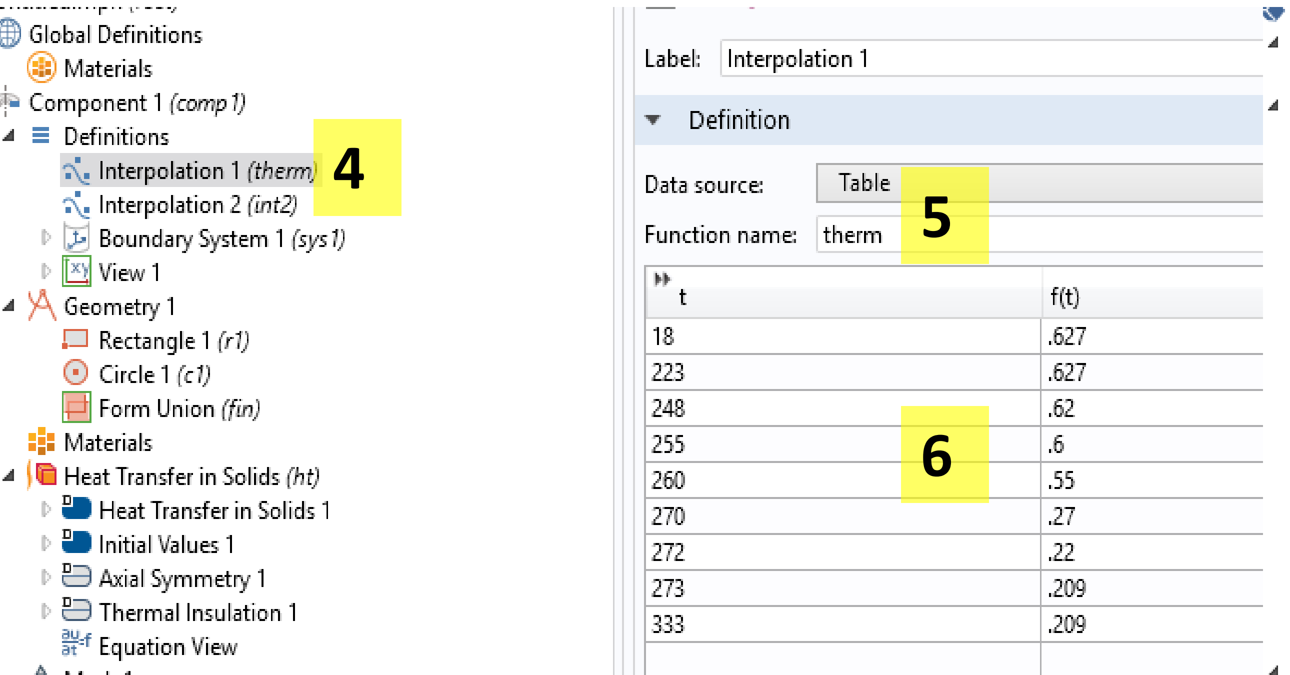



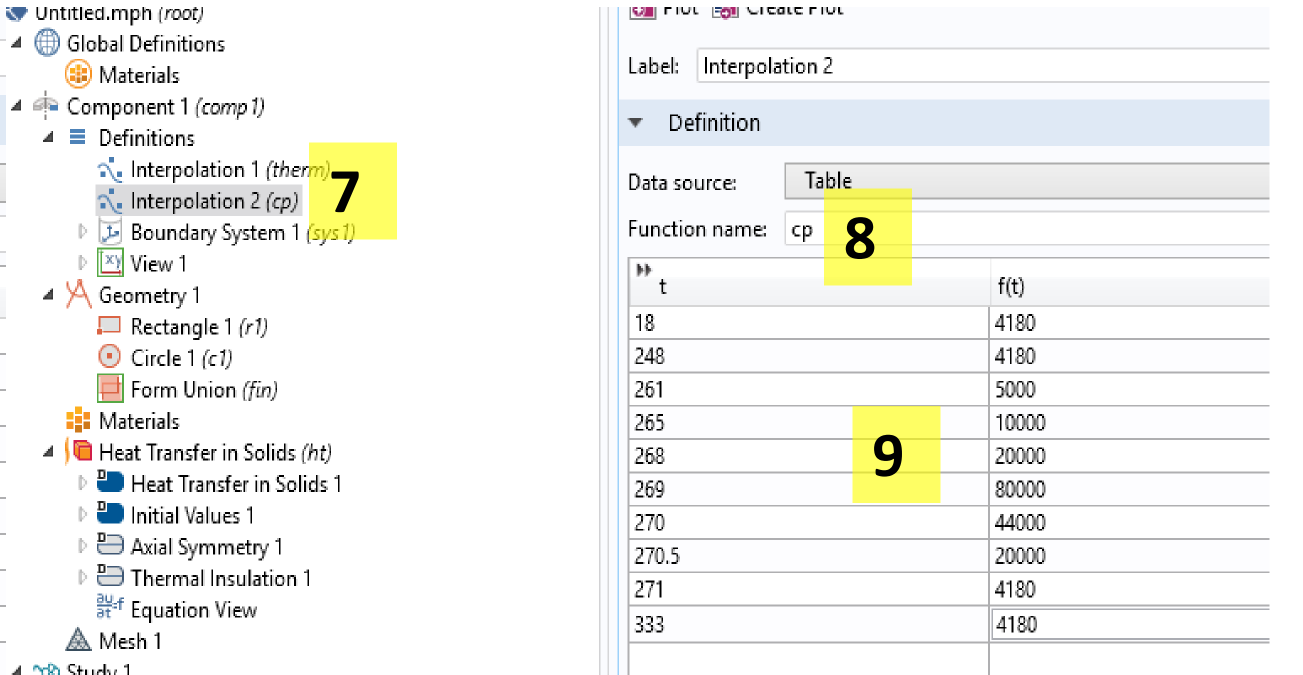



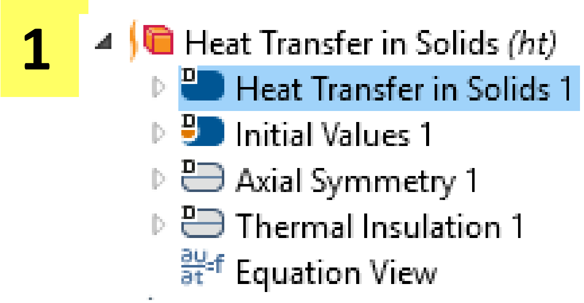





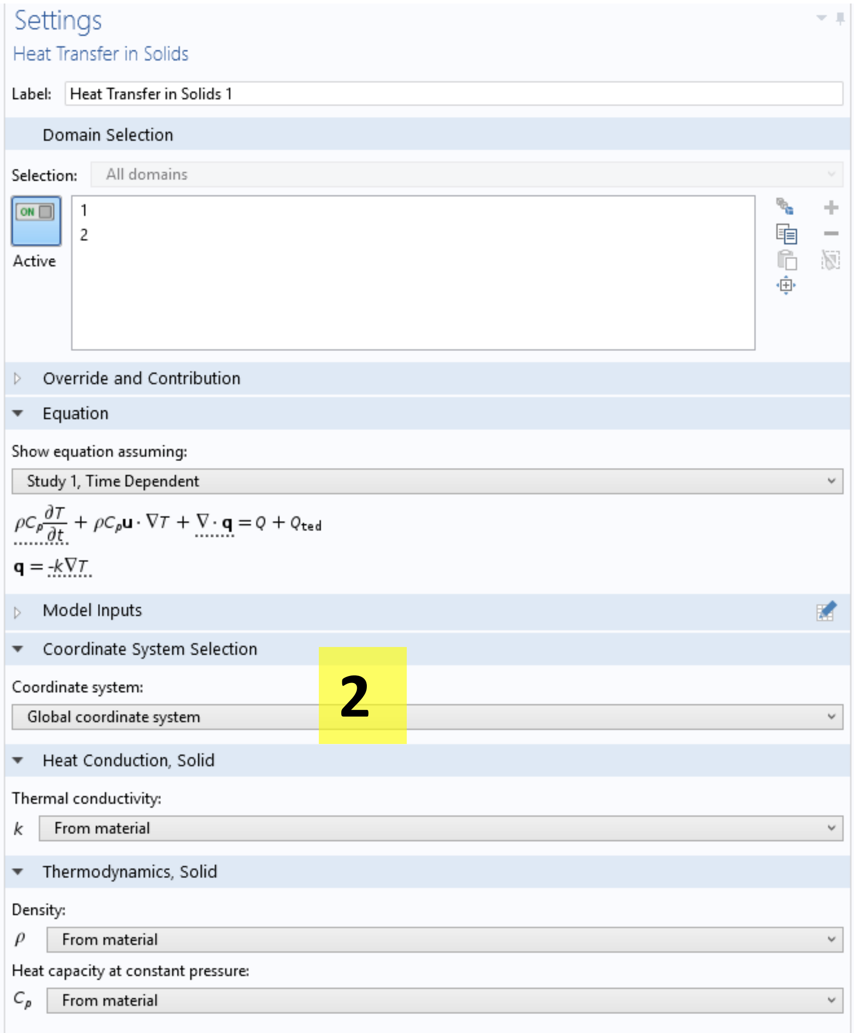

1) Right click Definitions
2) Select functions
3) Click ‘interpolation’. Do 1-3 again
4) Left click ‘interpolation 1’
5) Enter a function name
6) Enter the same data
7) Left click ‘interpolation 2’
8) Enter a function name
9) Enter the same data

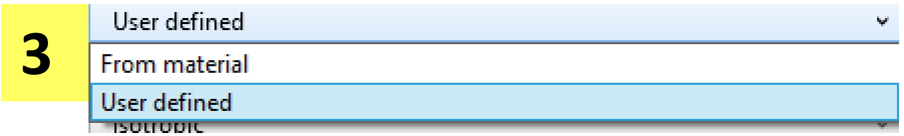


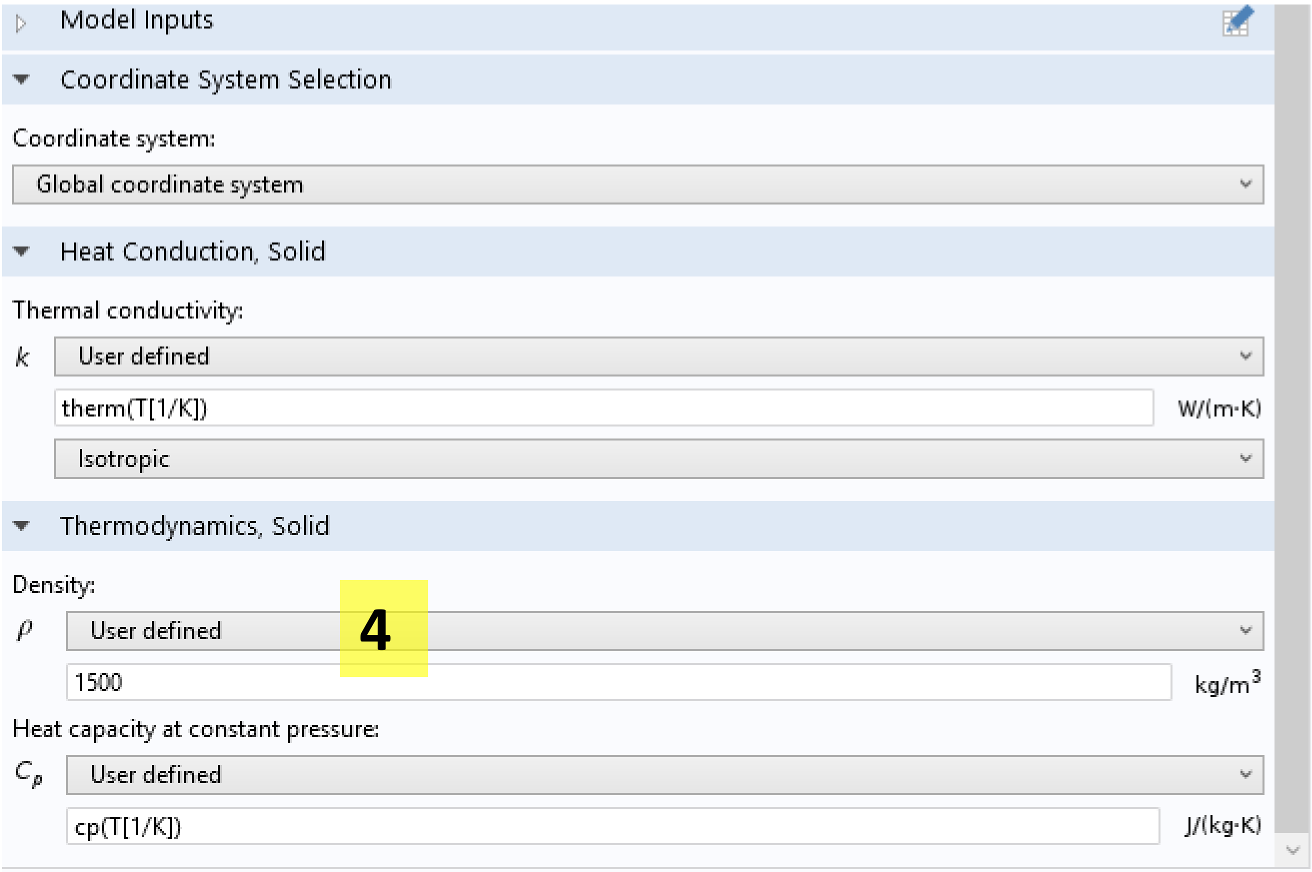



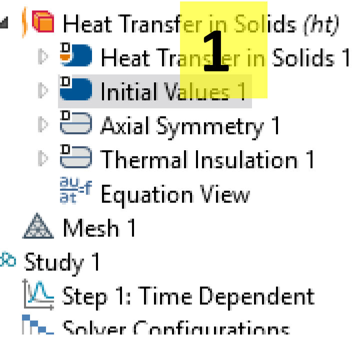


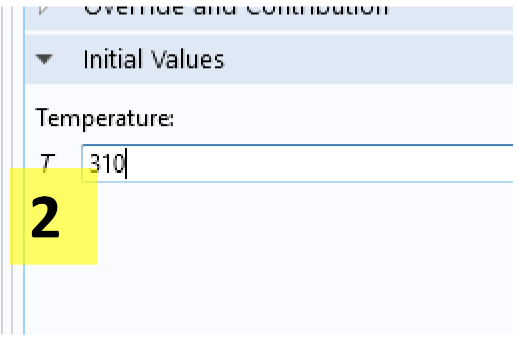

1) Go to ‘Heat transfer in solids’ >> ‘Heat transfer in solids 1’
2) In the middle panel, scroll down to k, ρ, and Cp
3) Select ‘user defined’ for all 3 properties
4) Input heat transfer properties as shown

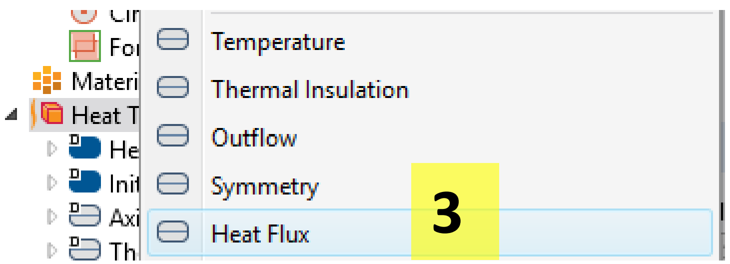


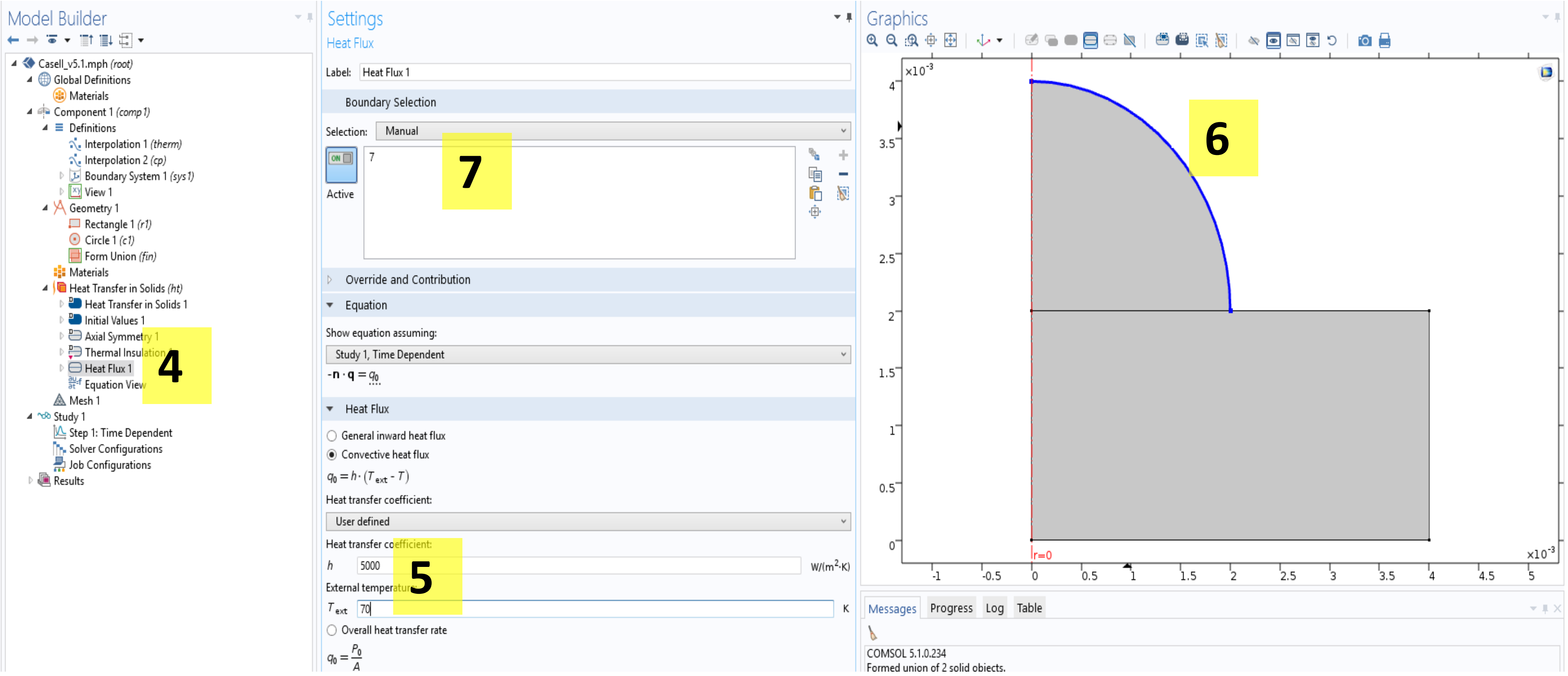





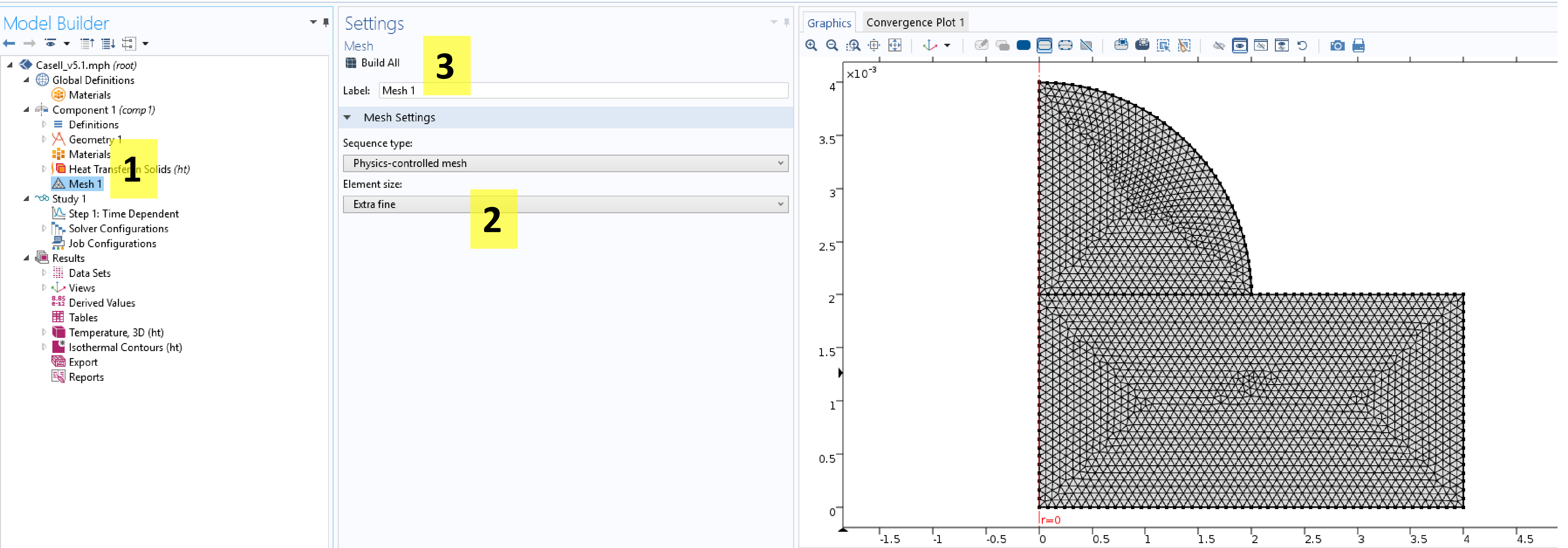



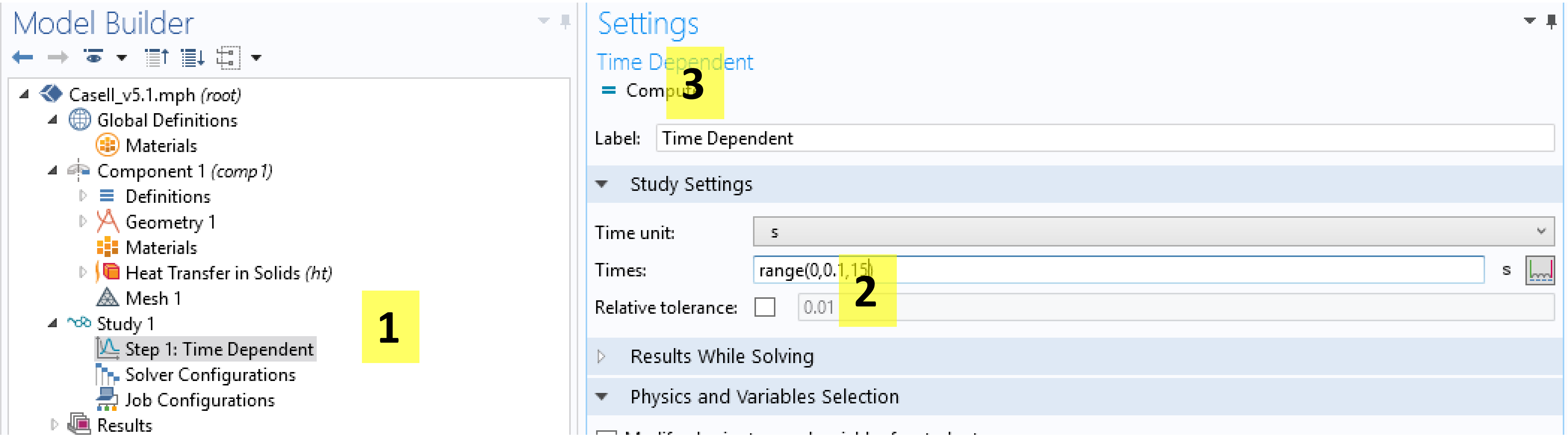



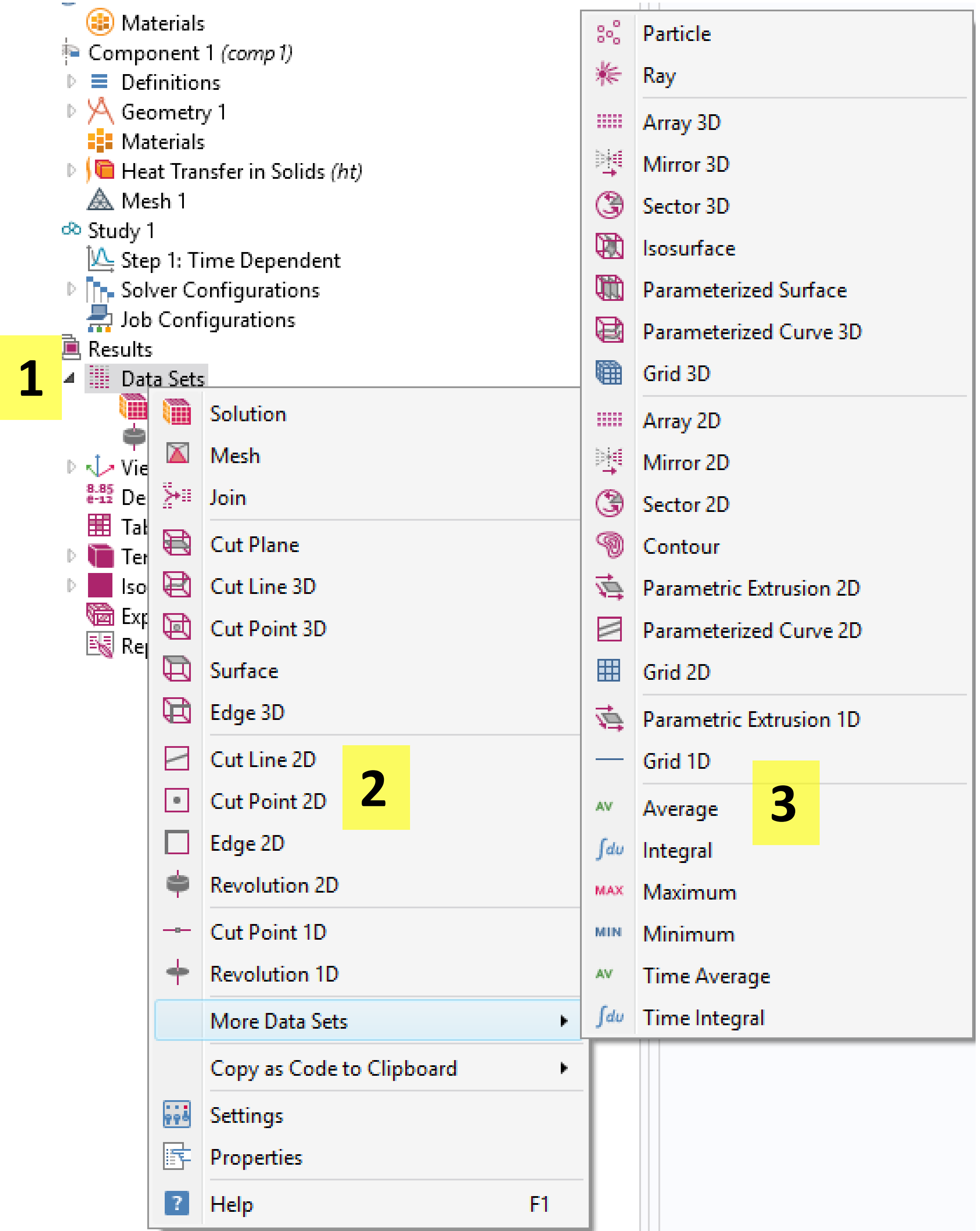

1) Left click ‘Initial values 1’
2) Enter 310 K
3) Right click ‘Heat transfer in solids’ and select ‘heat flux’
4) Left click on ‘heat flux 1’
5) Enter 5000 for h and 70 for T_ext
6) Left click the wart boundary
7) The number ‘7’ should appear

1) Left click mesh 1
2) Select ‘extra fine’
3) Click build all

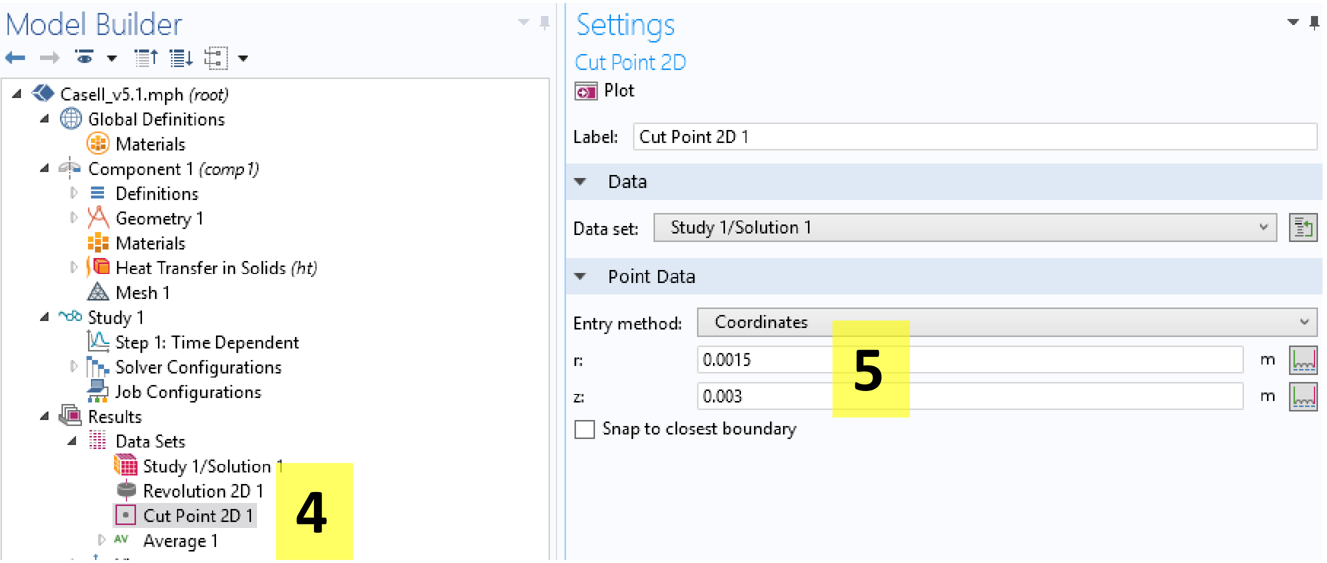


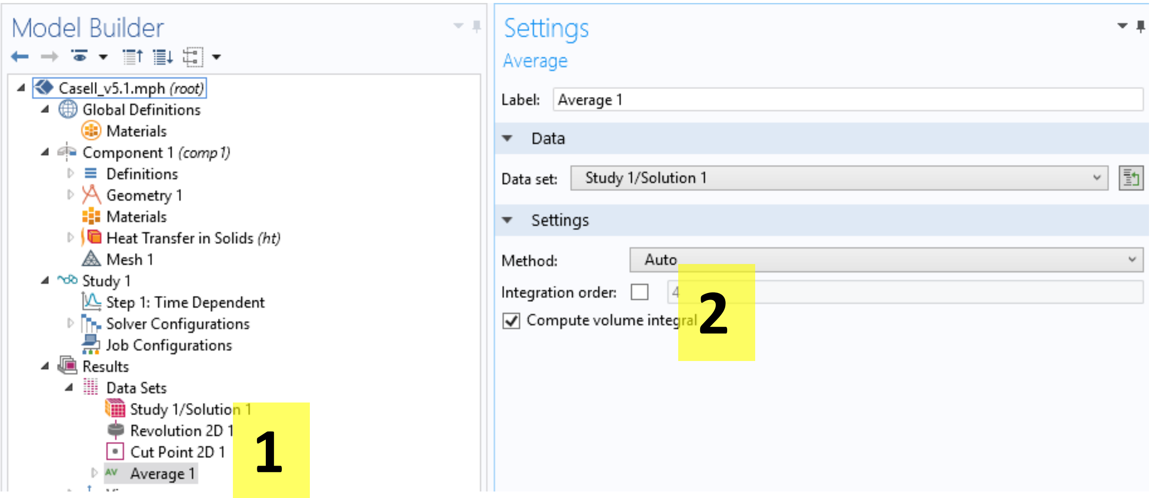
1) Left click “Step 1: Time dependent’
2) Enter range(0,.1,15)
3) Click compute

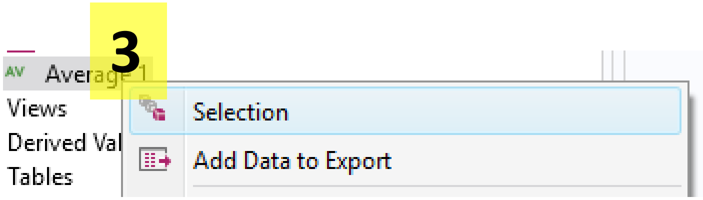


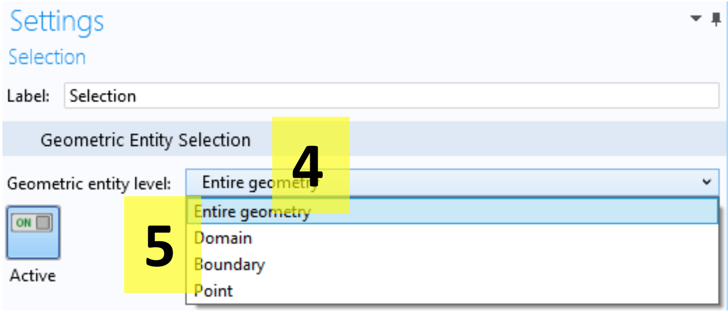


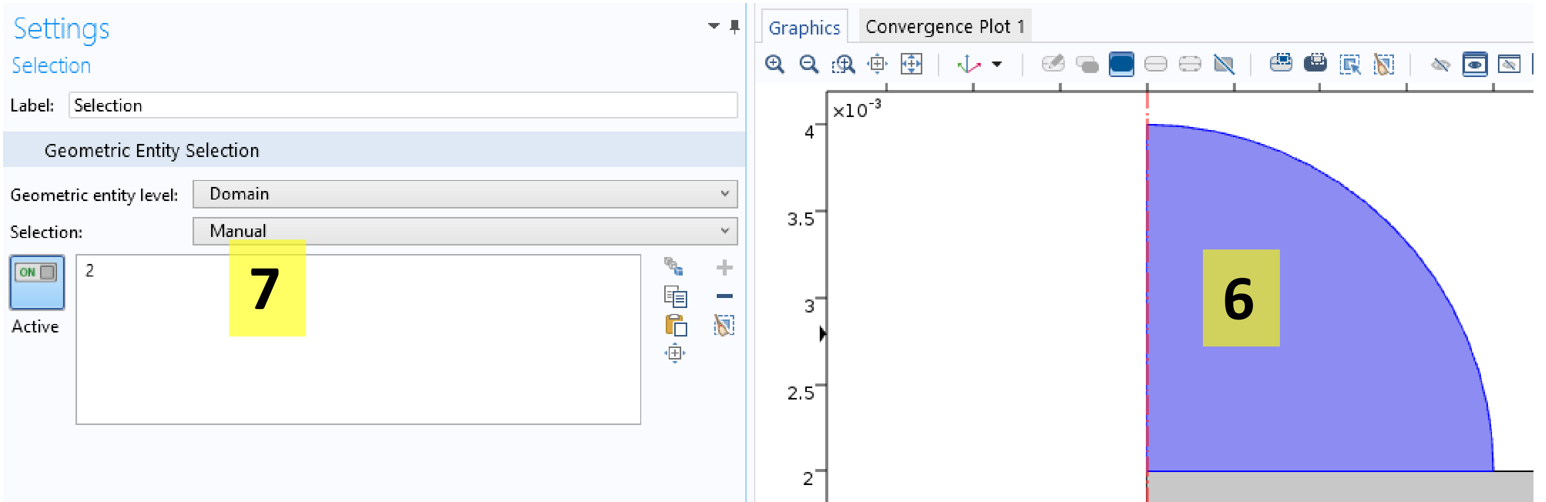

1) Right click data sets
2) Select ‘cut point 2d’
3) Do step 1 again and select ‘Average’
4) Left click ‘cut point 2D 1’
5) Enter r=0.0015 and z = 0.003



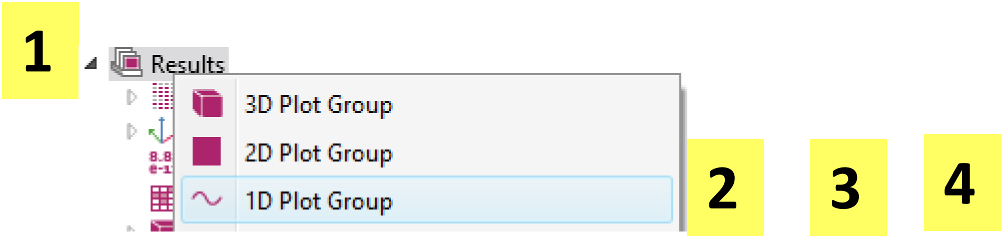


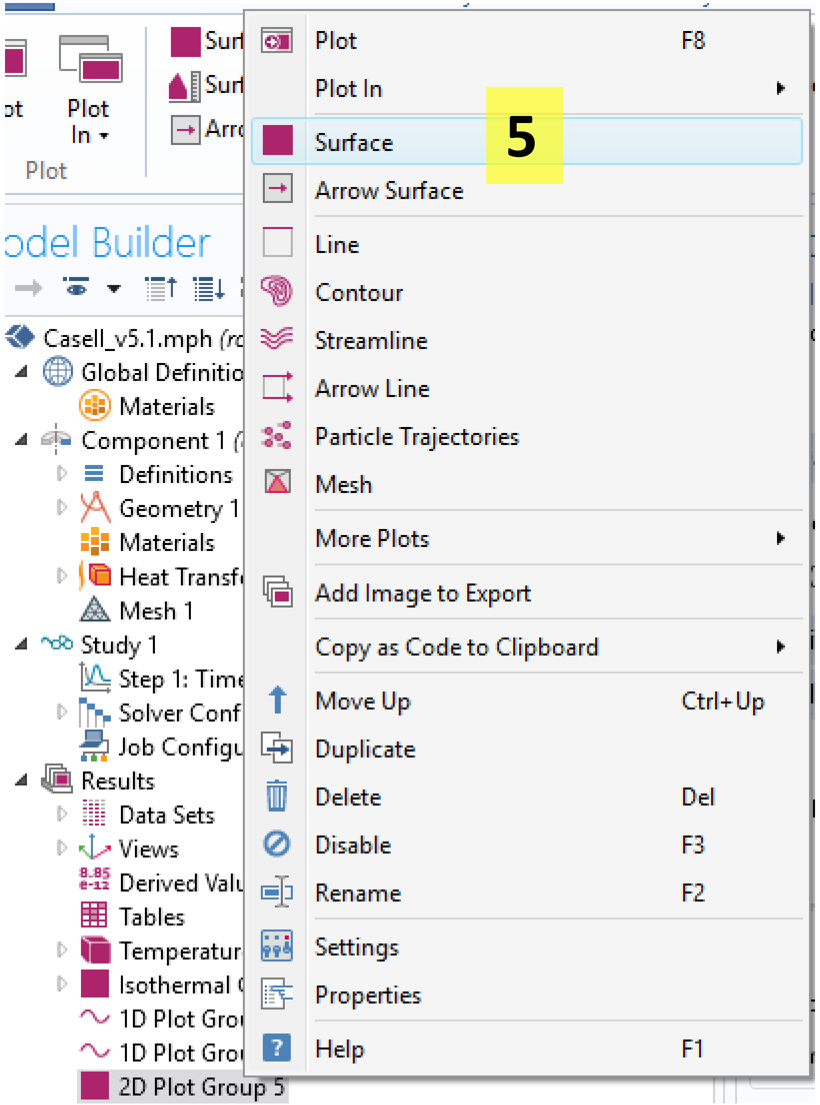



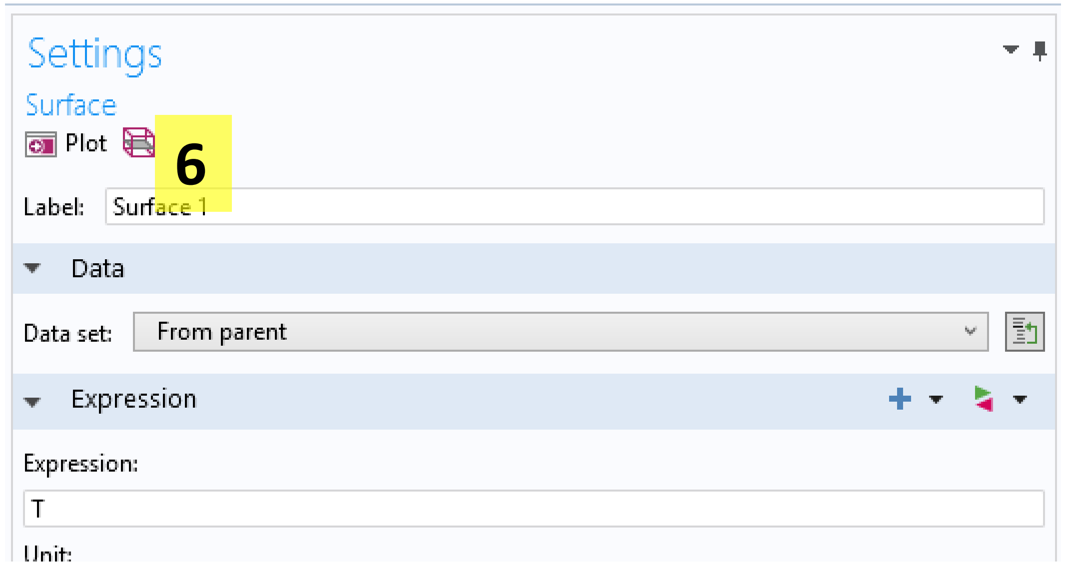



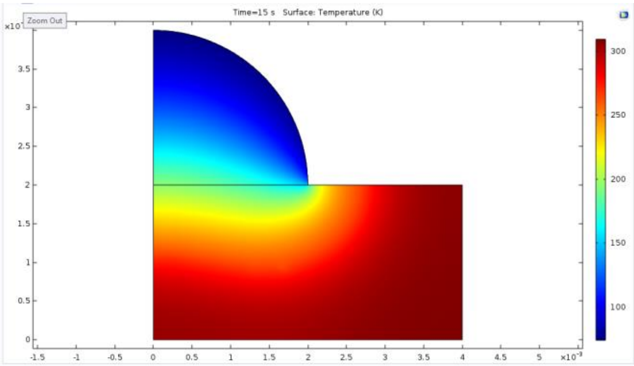

1) Left click “average 1”
2) Toggle Compute volume integral
3) Right click average 1 and pick ‘selection’
4) Click the dropdown
5) Select ‘Domain’
6) Left click the wort domain
7) You should see a ‘2’

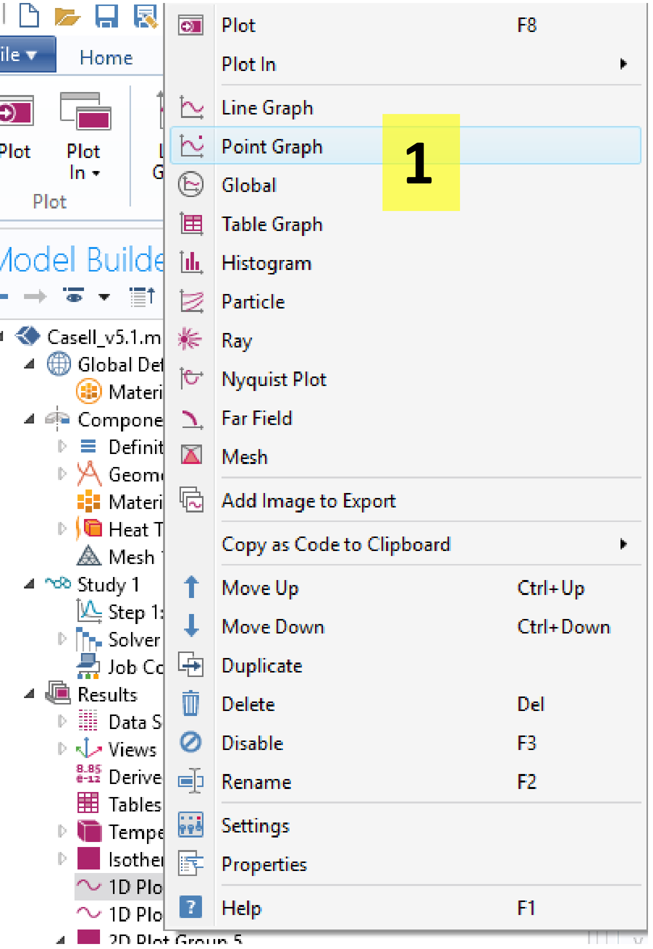


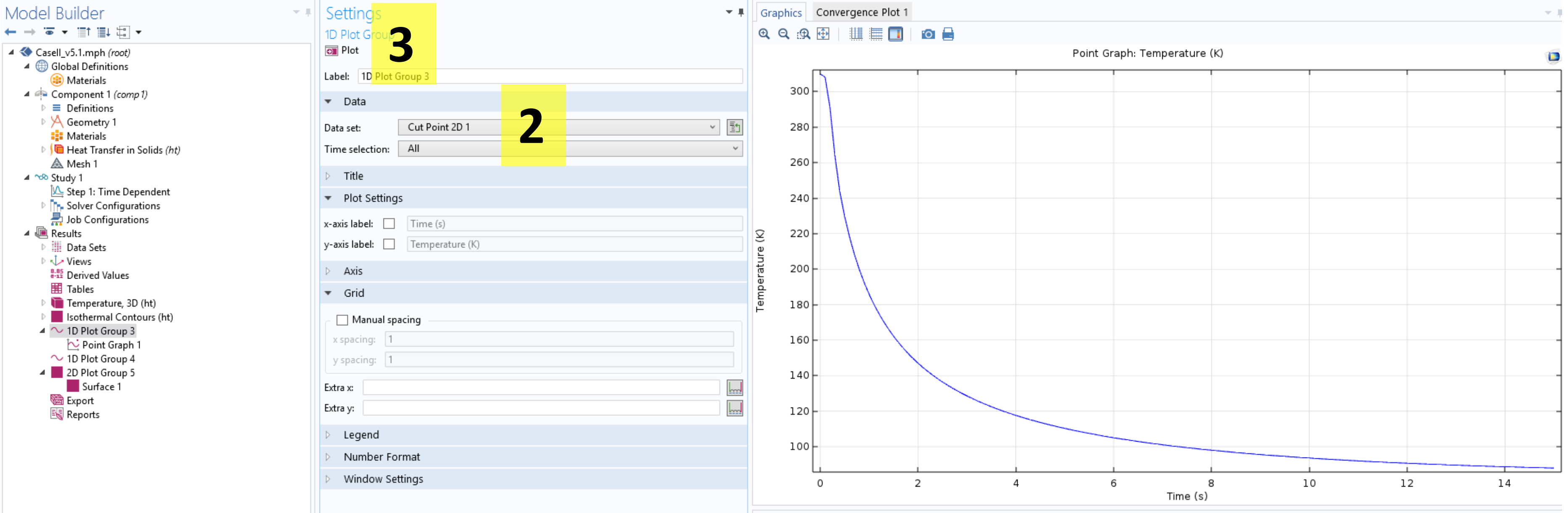

1) Right click 'results'
2) Select '1D plot group'
3) Repeat step 1 and Select '1D plot group'
4) Repeat step 1 and Select '2D plot group'
5) Right click 2D plot group 5 and select surface
6) Click plot



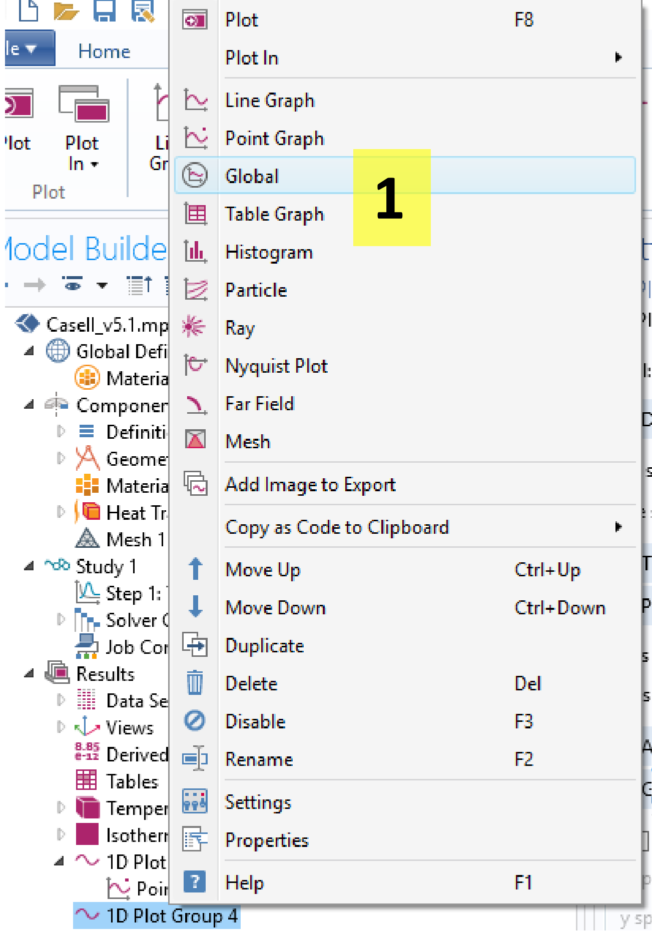



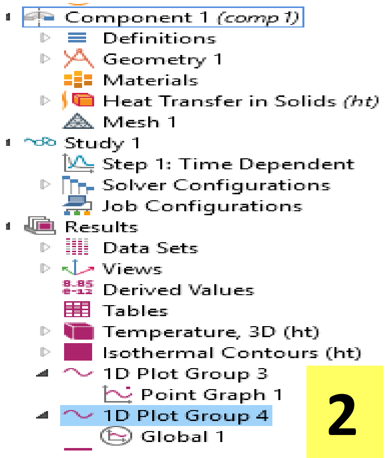


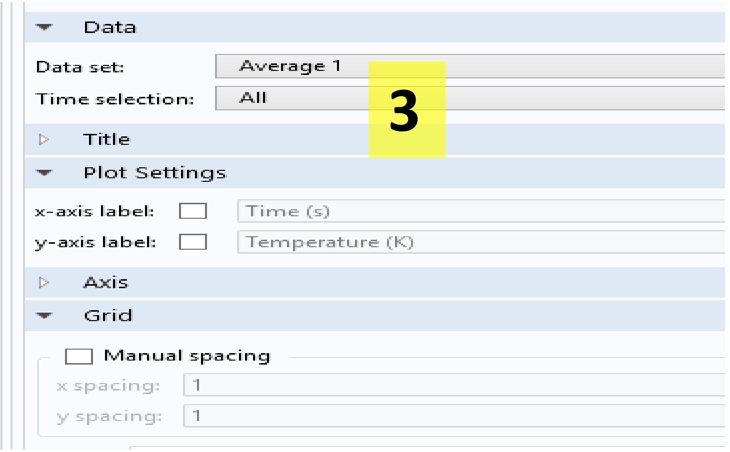

1) Right click 1D plot group 3 and select point graph
2) Select ‘cut point 2D 1’ from ‘data set’ dropdown
3) Click plot

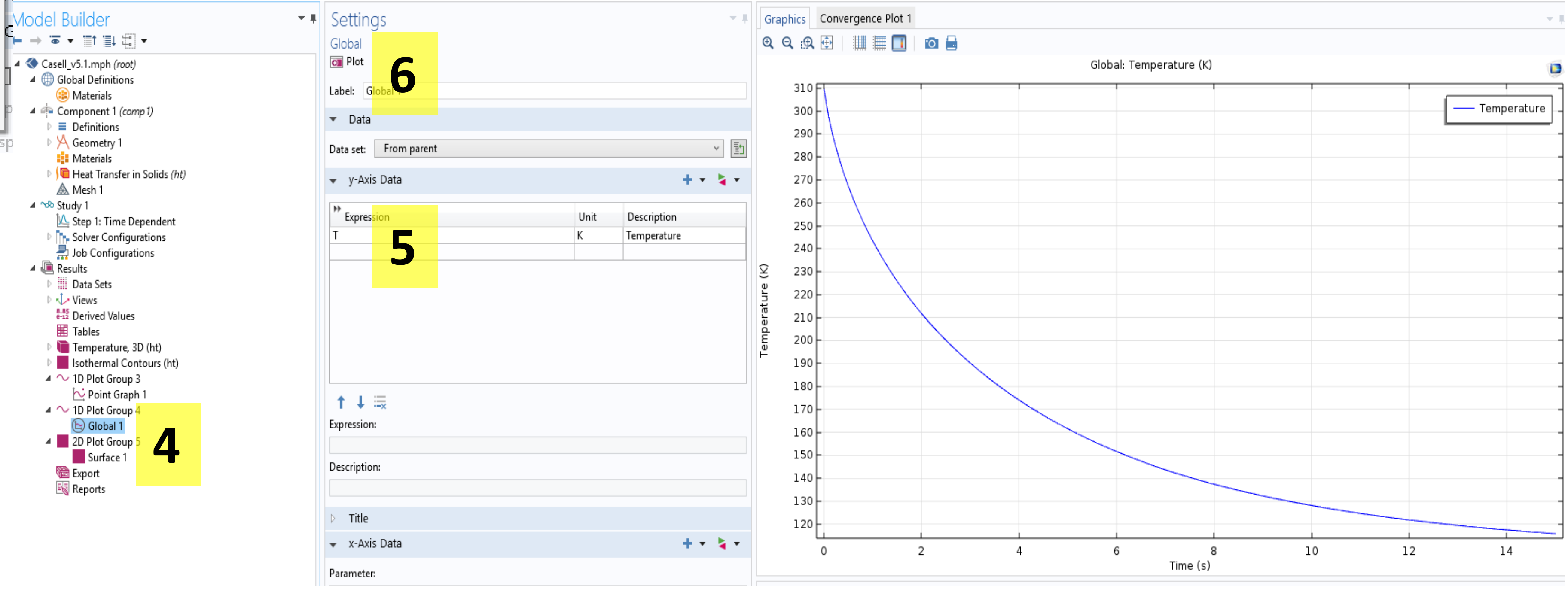


1) Right click 1D plot group 3 and select global
2) Left click '1D plot group 4'
3) Select 'Average 1'
4) Click 'global 1'
5) Enter T
6) Click plot



**Bee 4530/MAE 4530**

**Homework (Case studies I and II in text): Due Feb. 12, 2016 in class**

Go through the case studies and submit materials asked for. Each student needs to submit this individually. **All plots must be properly labeled (title and axis names)**. There are tutorials on the course website to help you use the software, please check "Tutorials" before asking.

**What to submit for Case study I:**

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
4) Make a surface plot of the temperature at t = 180s
5) Plot the temperature at the point (0.0075,0.01) as a function of time for
    5a) Probe temperature = 70, 90, and 110°C (consider using parametric sweep)
    5b) Plot in EXCEL (export data as a text file and import into EXCEL), label axes
6) Make a movie (*.avi file and *.gif file) of the temperature over time, using a surface plot. This **does not** have to be turned in.
7) The model is made in 2D, is there heat transfer in 1 or 2 directions? Explain.

**What to submit for Case study II:**

1) Write down the specific governing equation that is being solved
2) Provide the boundary conditions and initial conditions used
3) Submit a COMSOL plot of Heat Capacity versus Temperature
4) Submit a temperature surface plot at the final time AND comment on the region over which healthy cells are damaged
5) Plot average wart temperature versus time on its own plot.
6) Turn in a COMSOL plot of average temperature versus time and the cut point temperature vs time from the tutorial **on the same COMSOL plot** (hint: add a 1D plot group, right click the new plot group, add a point plot under it, right click it again and add a global plot. Then, click on the 'point graph' and change its data set to 'Cut point 2D 1. Click on the global 1 plot and change its data set to 'average 1'. This is why we have plots and plot groups)