Step 1: Problem Type Specification

Box 6.12
The only mode of heat transfer is by conduction. Therefore the problem is a transient conduction problem with no convection and heat source.

(1) Start COMSOL by double clicking on the icon on the Desktop.

(2) Select 2D axisymmetric under 'Select Space Dimension' tab.

(3) Click the blue 'Next' arrow next to the 'Select Space Dimension' title.

(4) Click left of 'Heat Transfer' to expand the option.

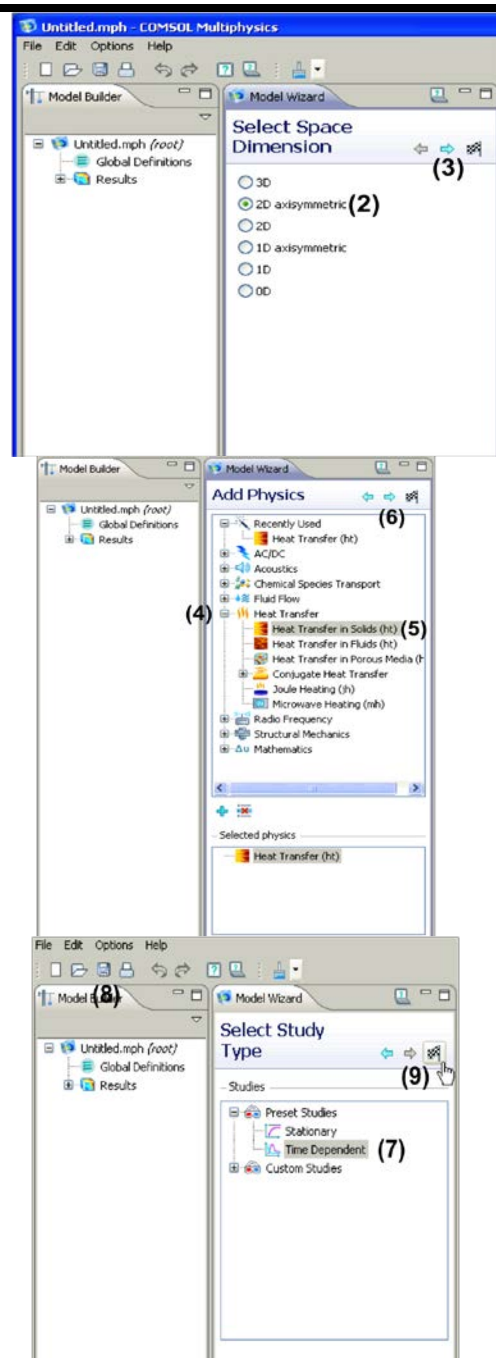
(5) Double click on 'Heat Transfer in Solids (ht)'.

(6) Click the blue 'Next' arrow next to the 'Add Physics' title.

(7) Left click on 'Time Dependent'.

(8) Click the 'Save' icon. Save often in order to prevent losing your work.

(9) Click on the race flag icon next to the 'Select Study Type' title.



Step 2: Geometry Creation

Box 6.13
The geometry in this case is a rectangle with a quarter-circle and is axisymmetric. The rectangle and the quarter-circle represent the skin and the wart, respectively. We will first draw the rectangle and then the quarter-circle.

Part A: Rectangle

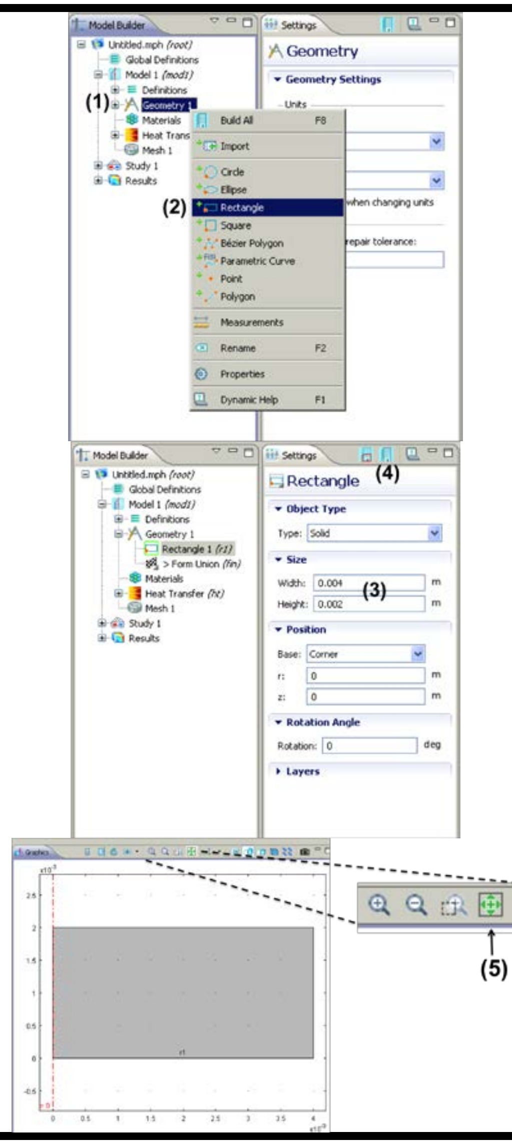
(1) Right click on ‘Geometry 1’.

(2) Select ‘Rectangle’ in the drop-down menu.

(3) Under the ‘Size’ tab, specify the width as 0.004m and height as 0.002m. These are the dimensions of the skin. Only half of the skin and the wart is considered as it is symmetric.

(4) Click on the ‘Build All’ icon at the top.

(5) Click on ‘Zoom Extents’ icon.



Part B: Quarter-circle

(1) Right click 'Geometry 1'.

(2) Select 'Bézier Polygon' from the drop-down menu.

(3) Click on 'Add Quadratic' button.

(4) In the edit fields provided under 'Control points', type in the following coordinates:

	r	z
1	0.002	0.002
2	0.002	0.004
3	0	0.004

(5) Click on 'Build All'.

(6) Click on 'Zoom Extents' to fit the geometry in the window.

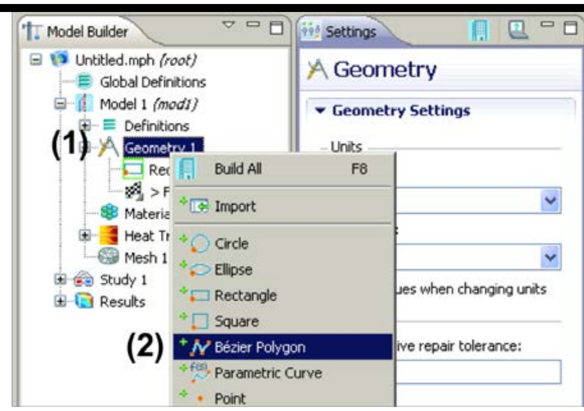
(7) Right click 'Geometry 1'.

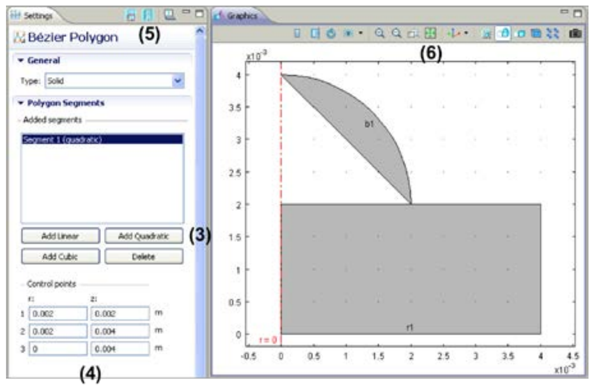
(8) Select 'Square'.

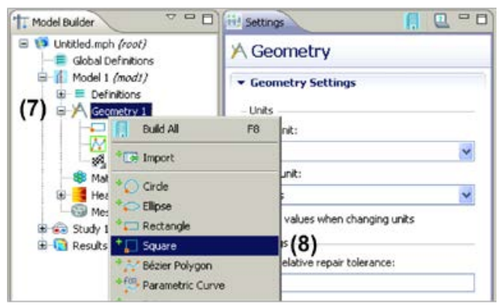
(9) Enter 0.002 for side length under 'Size' tab.

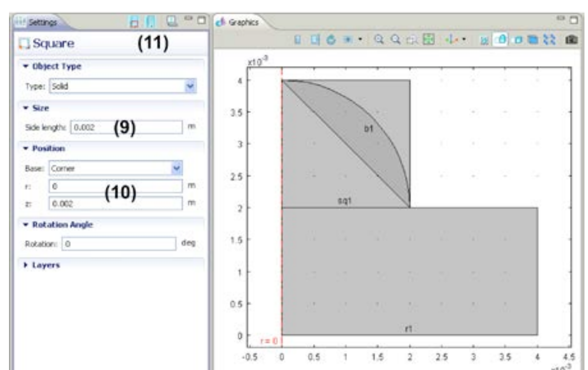
(10) Enter r = 0, z = 0.002 for corner under 'Position' tab.

(11) Click on 'Build All'.









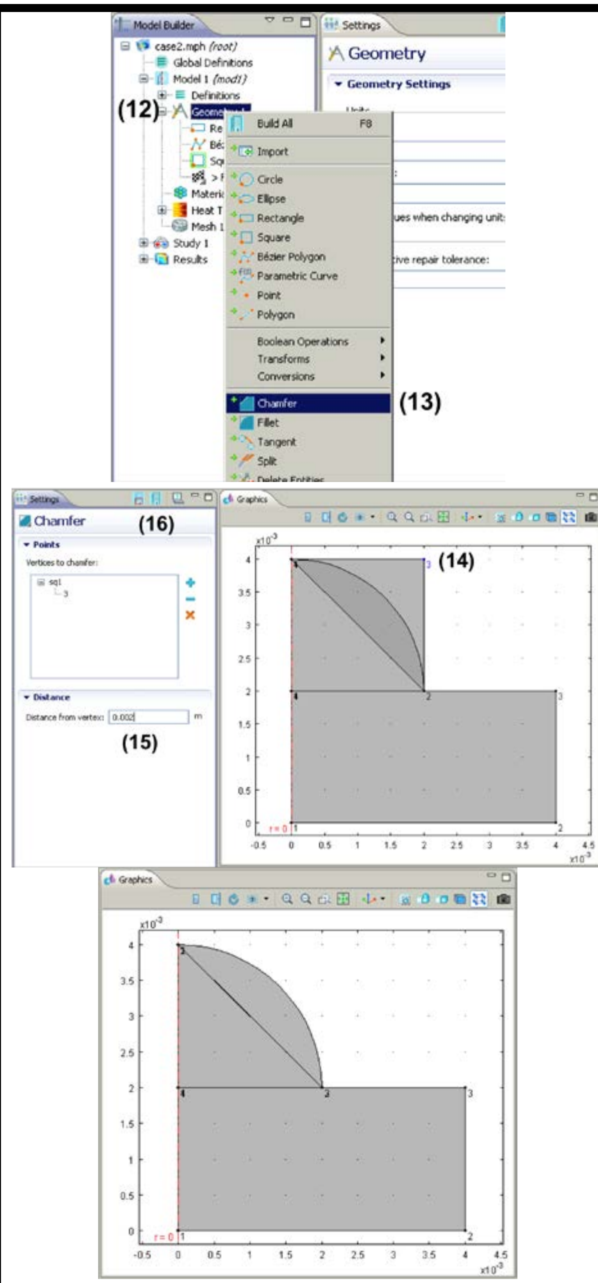
(12) Right click ‘Geometry 1’.

(13) Select ‘Chamfer’.

(14) Left click and then right click the vertex labeled ‘3’ (the upper-right corner of the square). The vertex should turn red upon left clicking and then blue upon right clicking. When the vertex has been successfully added, it should say *sq1 > 3* under the ‘Vertices to chamfer:’ box.

(15) Type in 0.002m for ‘Distance from vertex’.

(16) Click on ‘Build All’.



(17) Right click ‘Geometry 1’.

(18) Select Boolean Operations > Union.

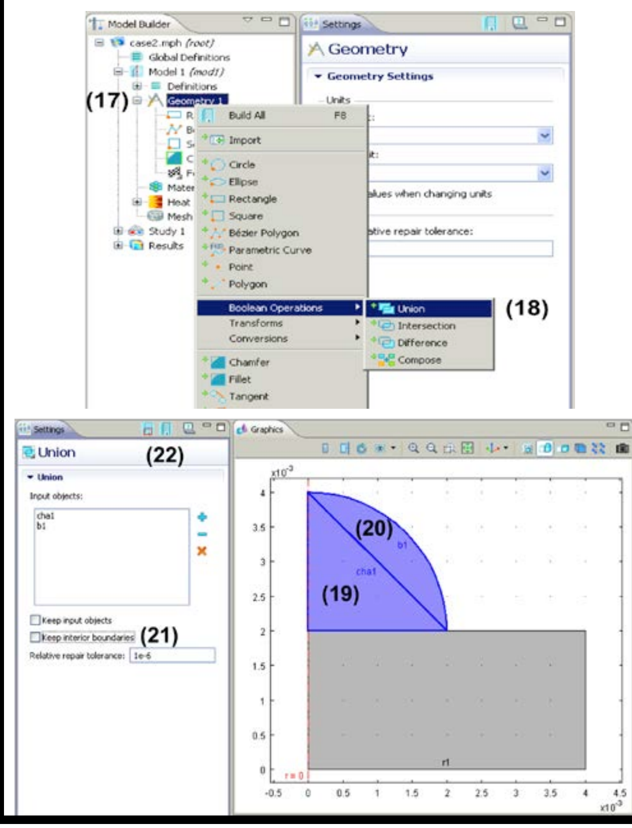
(19) Left click, then right click in the middle of the triangle.

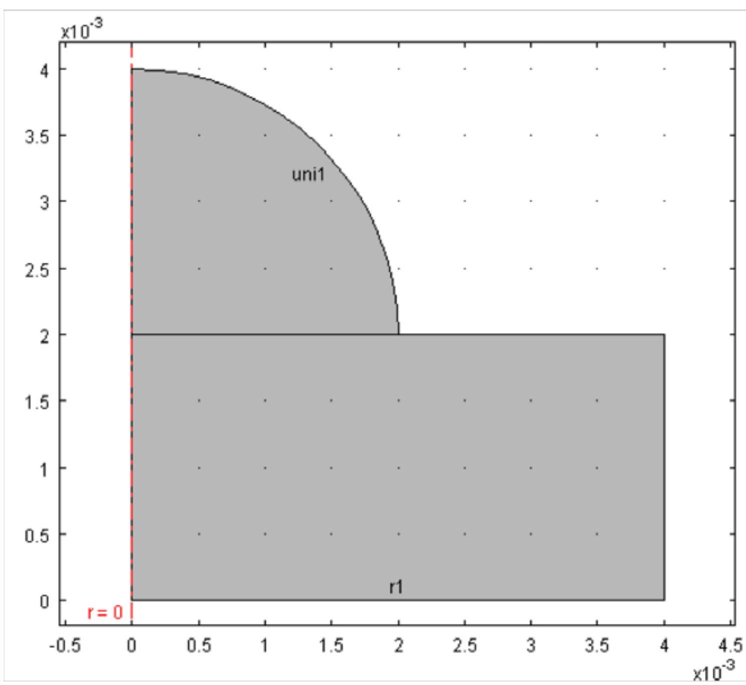
(20) Left click, then right click in the middle of the Bézier Polygon.

(21) Uncheck ‘Keep interior boundaries.’

(22) Click on ‘Build All’.

Your figure should now look like the following figure below.





Step 3: Meshing

Box 6.14

Meshing is dividing the geometry into small elements. We will create meshes with the same mesh density for both the skin and the wart faces.

(1) Right click on ‘Mesh 1’.

(2) Select ‘Free Triangular’.

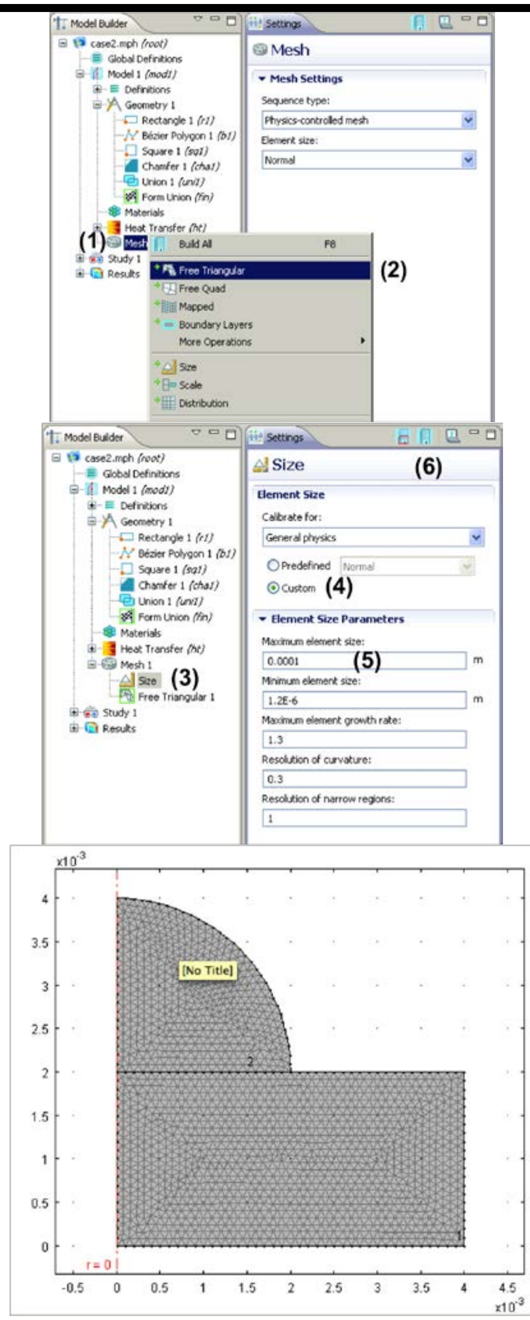
(3) Select Mesh 1 > Size.

(4) Select ‘Custom’ in the ‘Element Size’ section.

(5) Change the maximum element size to 0.0001m.

(6) Click on ‘Build All’.

Your mesh should now look like the figure to the right.



Step 4: Material Properties and Initial Conditions

Box 6.15

We are solving for the heat transfer equation. The material properties required for the analysis are, therefore, thermal conductivity, specific heat, and density. The density of the normal tissue layer is assumed to be 1000 kgm^{-3} and that of the wart as 1500 kgm^{-3}. The thermal conductivity and specific heat constant are both taken to be functions of temperature, as specified in Figure 6.3. The temperature inside the skin and the wart is 37°C (=310K) initially.

(1) Right click on 'Heat Transfer'.

(2) Select 'Heat Transfer in Solids' from the menu.

(3) Left click on 'Heat Transfer in Solids 1'.

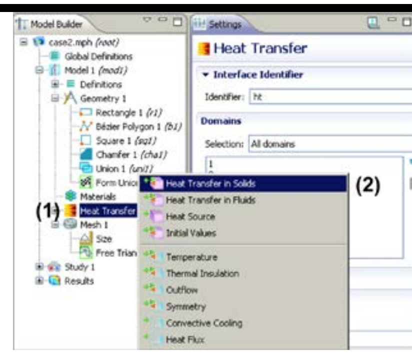
(4) After selecting 'User defined' for k, ρ, Cp, enter therm_cond(T) for k, 1000 for ρ, and heat_cap(T) for Cp.

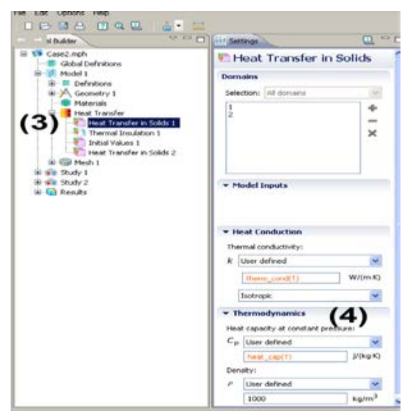
(5) Left click on 'Heat Transfer in Solids 2'.

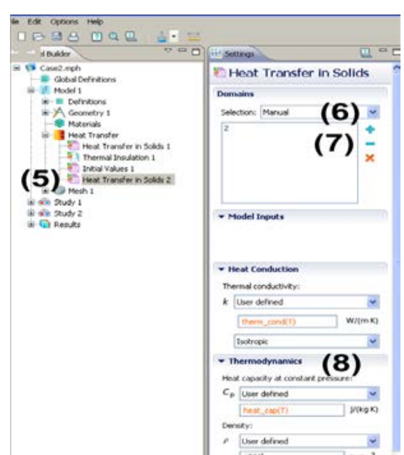
(6) In the Select drop-down menu, select 'All domains'.

(7) Left click '1', and click the '-' sign.

(8) After selecting 'User defined' for k, ρ, Cp, enter therm_cond(T) for k, heat_cap(T) for Cp, and 1500 for ρ.







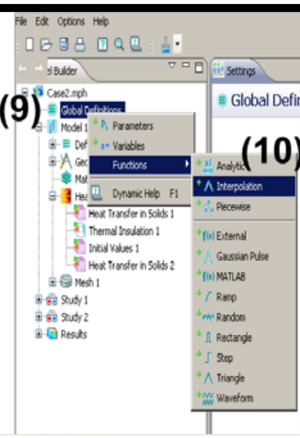
(9) Right click on 'Global Definitions'.

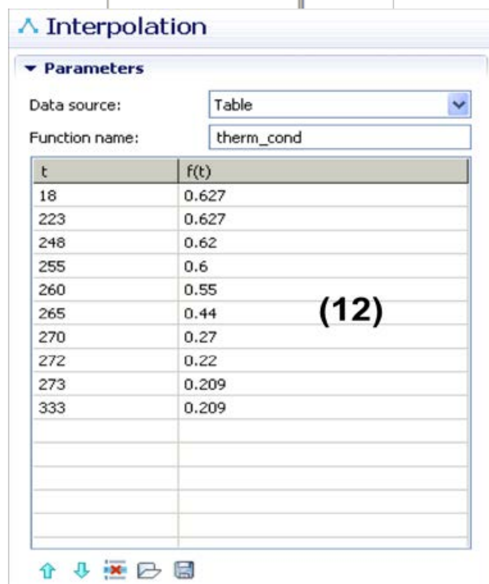
(10) Select Functions > Interpolation.

(11) Repeat (9) & (10).

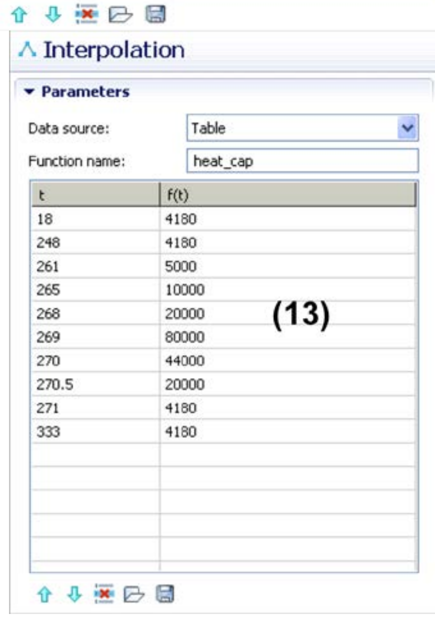
(12) For Interpolation 1, enter the function therm_cond(T) as shown.

(13) For Interpolation 2, enter the function heat_cap as shown.





t	f(t)
18	0.627
223	0.627
248	0.62
255	0.6
260	0.55
265	0.44
270	0.27
272	0.22
273	0.209
333	0.209



t	f(t)
18	4180
248	4180
261	5000
265	10000
268	20000
269	80000
270	44000
270.5	20000
271	4180
333	4180

Step 5: Boundary Conditions

Box 6.16

The boundary conditions for the problem are shown in Figure 6.2(b). The curved surface of the wart has a convective boundary condition and all other boundaries have zero heat flux condition (insulated boundaries). The default boundary condition in COMSOL is insulation, so we need to specify boundary condition for only the curved surface.

(1) Right click on 'Heat Transfer (ht)'.

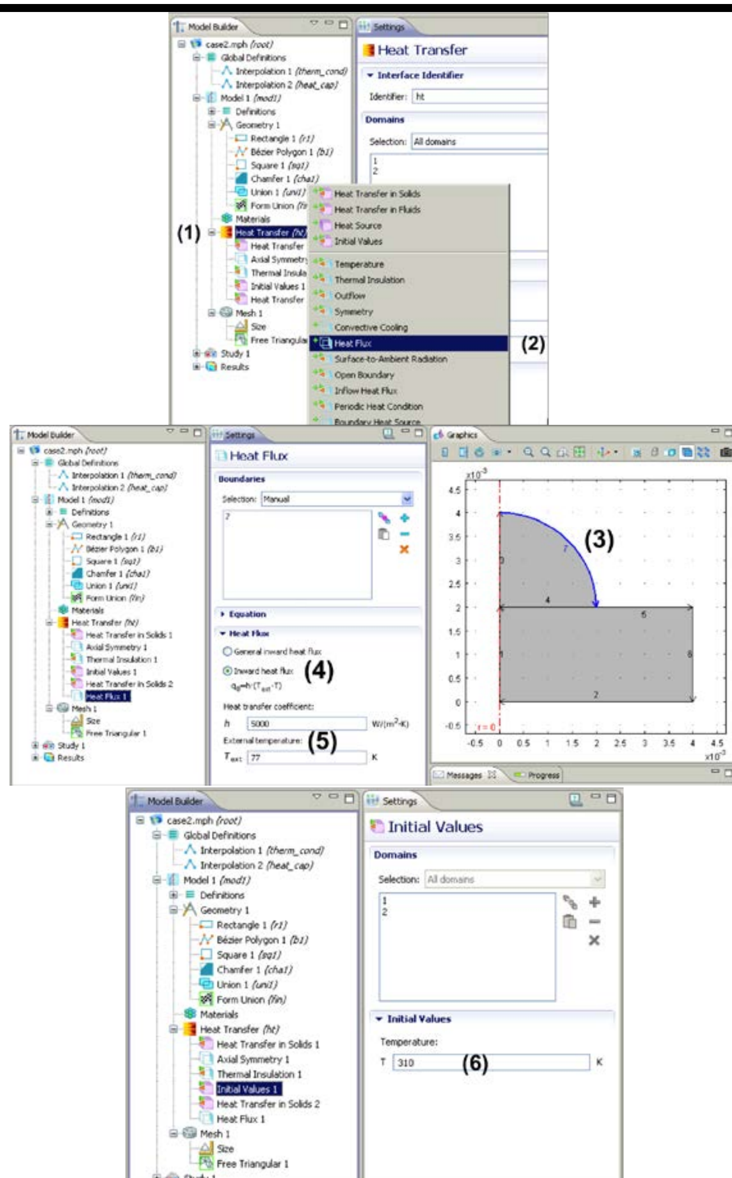
(2) Select Heat Flux from the menu.

(3) Left click, and right click on boundary (7).

(4) Select Inward heat flux.

(5) Enter 5000 for h and 77 for $T_{ext.}$

(6) Select Heat Transfer (ht) > Initial Values 1 and enter 310K in the Temperature edit field.



Step 6: Specifying Solver Parameter

Box 6.17

We need to specify the time interval of the process as well as the times at which the solution will be stored by the solver. We will solve the problem for 15s. as you can see from Figure 6.3, the thermal properties change very rapidly near 0°C. Therefore to obtain an accurate solution, the time steps taken by the solver should be small so that the temperature dependence of thermal conductivity and specific heat is resolved precisely. By default, the solver calculates the time step size internally. In this example, we will force it to take smaller time steps.

(1) Click left of 'Study 1' to expand it.

(2) Select 'Time Dependent 1'.

(3) Enter range(0,0.1,15)

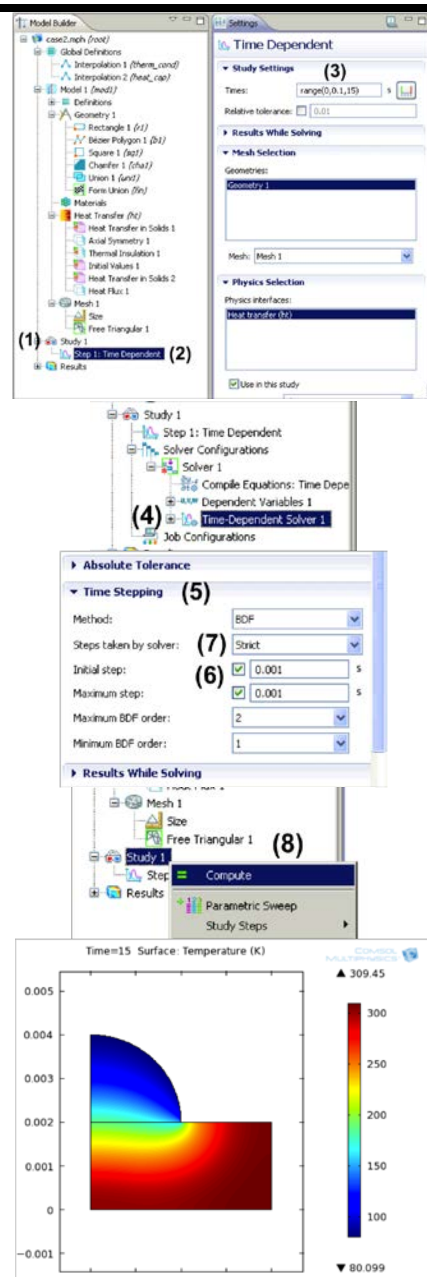
(4) Click on Study 1 > Solver Configurations > Solver 1 > Time-Dependent Solver 1.

(5) Expand the Time Stepping tab.

(6) Check the boxes for Initial step and Maximum step and enter 0.001 for both of them.

(7) Open the drop-down menu for Steps taken by solver and choose Strict.

(8) Right click on 'Study 1' and select Compute.



Step 7: Postprocessing

<u>Plotting temperature versus time at wart interior</u>

We will now plot the temperature history at point (0.0015, 0.003) in the interior of the wart, to see how the temperature varies with time at that location.

(1) Right click Data Sets.

(2) Select Cut Point 2D.

(3) Enter the point r = 0.0015, z = 0.003.

(4) Right click Results.

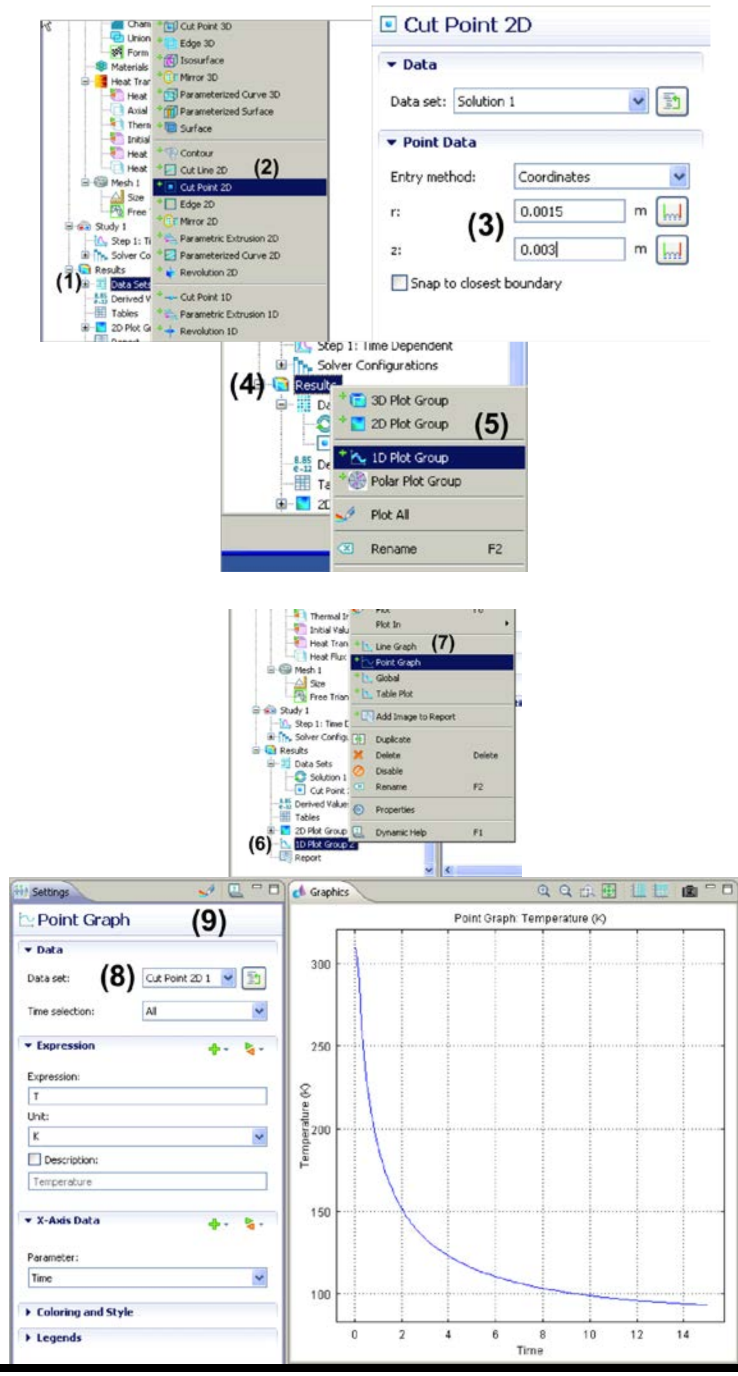
(5) Select 1D Plot Group.

(6) Right-click on Results > 1D Plot Group 2.

(7) Select Point Graph.

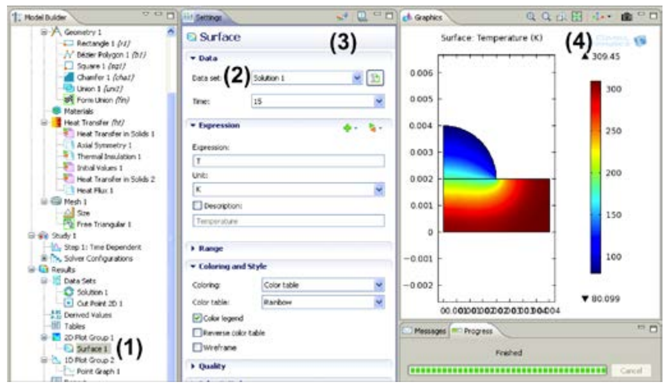
(8) Select Cut Point 2D 1 from Data set drop-down menu.

(9) Click on the 'plot' icon.



Obtaining the surface plot at a specific time

We now plot the temperature contour in the slab at t=15s.

(1) The surface plot is automatically generated under 2D Plot Group > Surface 1. (2) Select 'Solution 1' under Data Set. (3) Click the 'plot' icon if needed. (4) If the picture is too small or not there, try clicking the zoom extent icon.	

Step 8: Optimization

We will now try to optimize the cryosurgery process using computations. Optimization has been discussed in Section 5.5. We will implement the objective function defined in that section to determine the optimum time for cryosurgery.

<u>Step 1. Select domains</u>

(1) Right-click on Model 1 *(mod1)* > Definitions.

(2) Select Model Couplings > Integration.

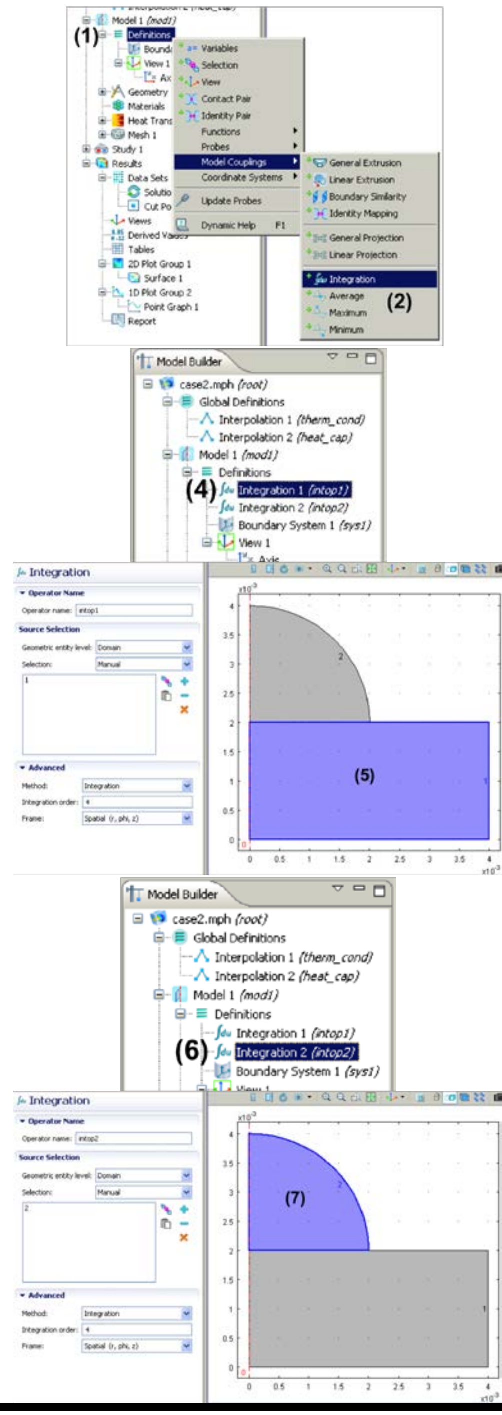
(3) Repeat step (1)&(2) one more time.

(4) Select Model 1 *(mod1)* > Definitions > Integration 1 *(intop1)*.

(5) Left click, then right click on subdomain 1.

(6) Select Model 1 *(mod1)* > Definitions > Integration 2 *(intop2)*.

(7) Left click, then right click on subdomain 2.



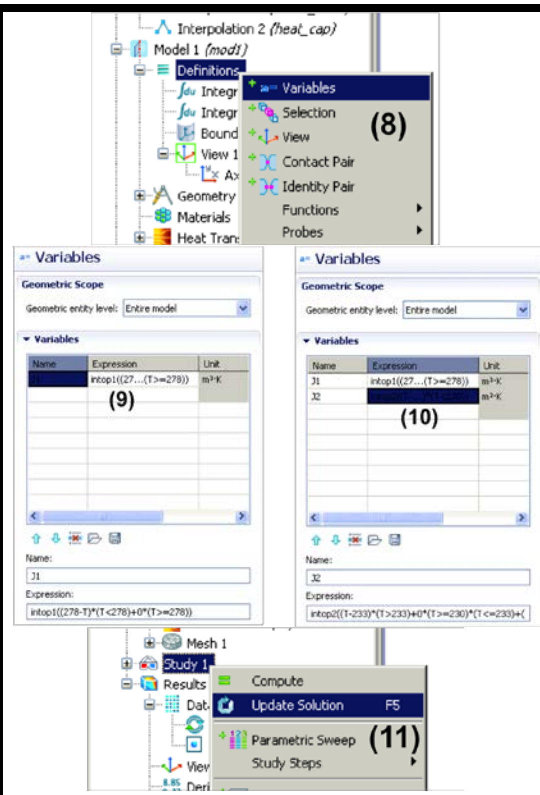
Step 2. Define objective function

(8) Right click on Model 1 *(mod1)* > Definitions and select 'Variables'.

(9) Click on the first row, and type in the following:
Name: J1
Expression: intop1((278- T) * (T<278) + 0*(T>=278))

(10) Click on the second row, and type in the following:
Name: J2
Expression: intop2((T-233) * (T>233) + 0*(T>=230) * (T<=233)+(230-T) * (T<230))

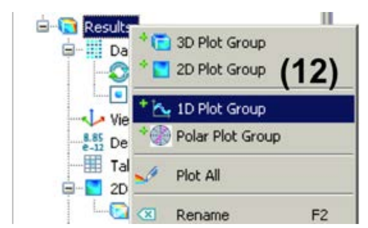
(11) Right click on Study 1 and select 'Update Solution'.

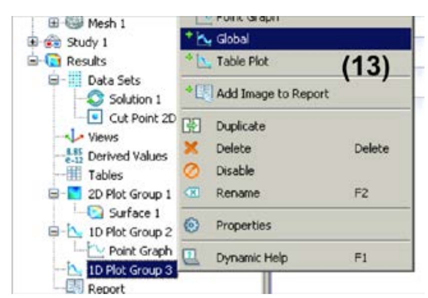


Step 3. Plot objective function versus time

(12) Right click on Results and select 1D Plot Group.

(13) Right click on Results > 1D Plot Group 3 and select Global.





(14) In the Expression edit field, type in J1+J2

(15) Click on the 'plot' icon.

The resulting figure is the objective function obtained as function of time. It can be observed that the objective function is minimum close to 2s.