Step 1: Problem Type Specification

(1) Open COMSOL Multiphysics.

(2) Select 2D Axisymmetric under Select Space Dimension.

(3) Click on blue arrow.

(4) Click on Heat Transfer > Heat Transfer in Solids (ht)

(5) Click on '+' icon.

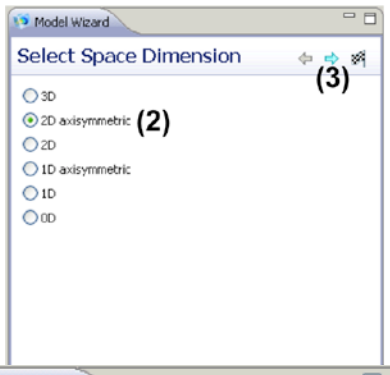
(6) Click on Chemical Species Transport > Transport of Diluted Species (chds)

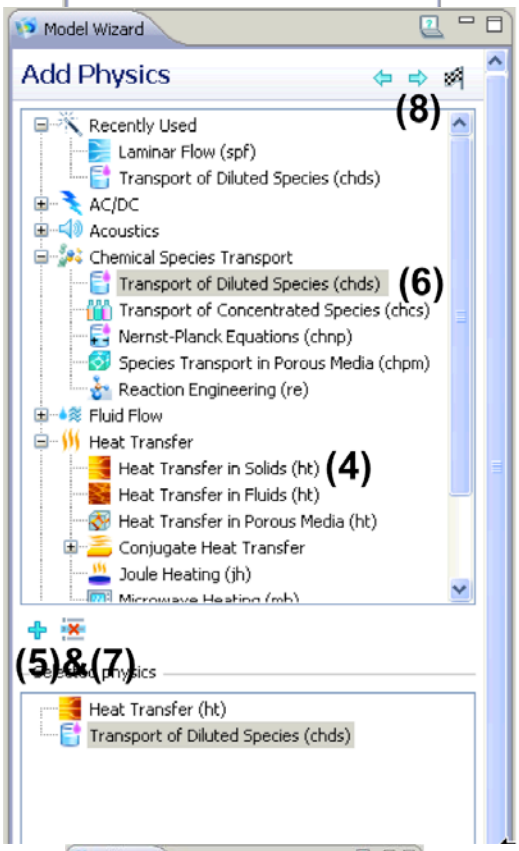
(7) Click on '+' icon.

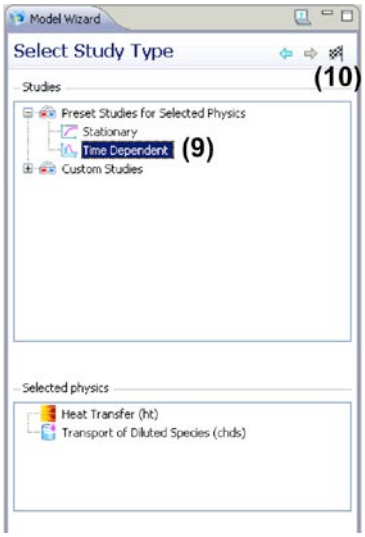
(8) Click on blue arrow.

(9) Click on Time Dependent

(10) Click on race flag icon.







Step 2: Geometry Creation

(1) Right click on Geometry 1 and select Rectangle.

(2) Enter the following information

Width (m)	Height (m)	Base	r	z
7e-3	3.425e-3	Corner	0	0

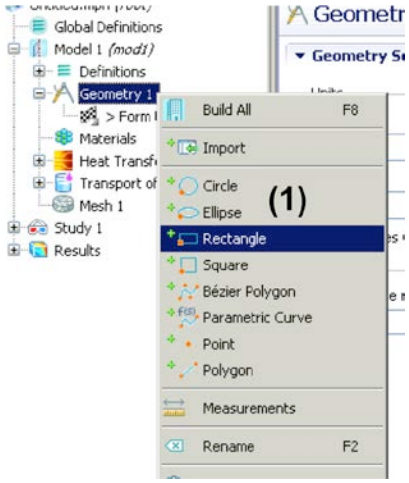
(3) Click on Build All icon.

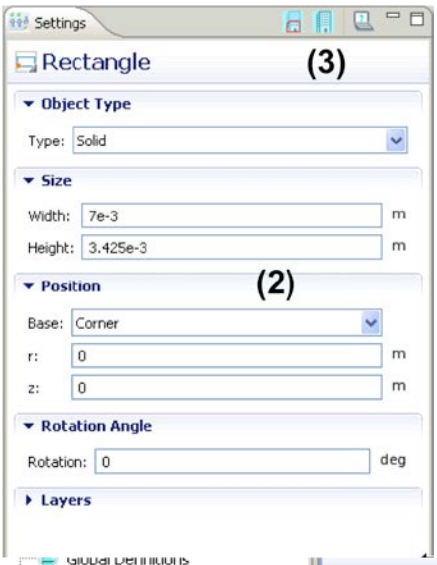
(4) Right click on Geometry 1 and select Rectangle.

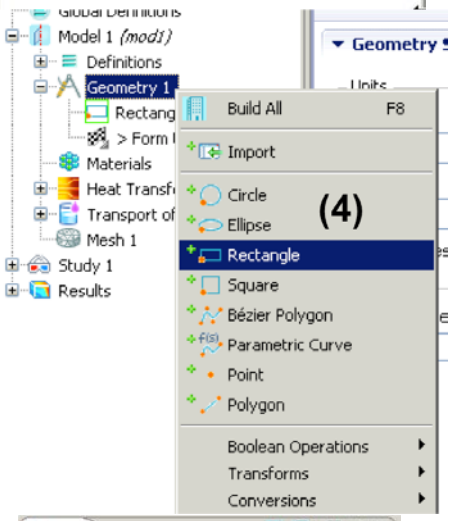
(5) Enter the following information

Width (m)	Height (m)	Base	r	z
7e-3	1.5e-3	Corner	0	3.425e-3

(6) Click on Build All icon.







(7) Repeat (4), (5), (6) with the following specifications:

Width	Height	Base	r	z
7e-3	75e-6	Corner	0	4.925e-3

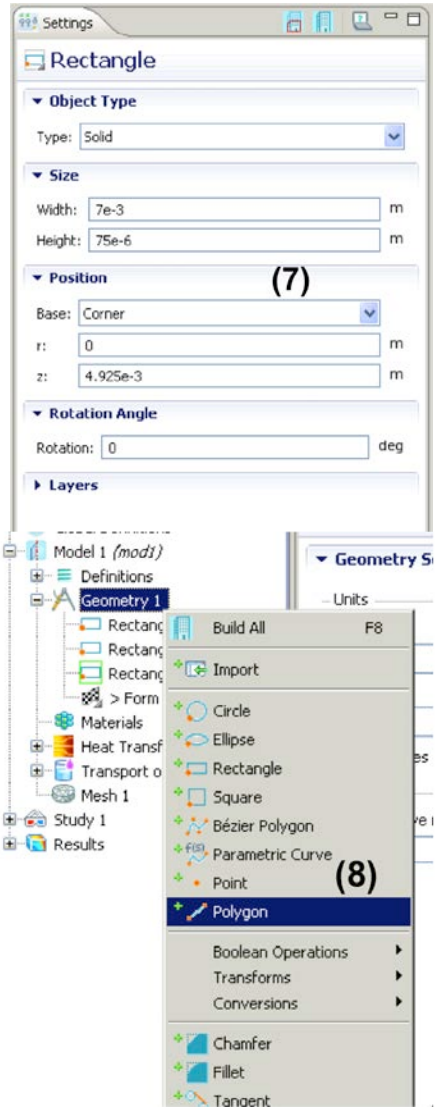
(8) Right click on Geometry 1 and select Polygon.

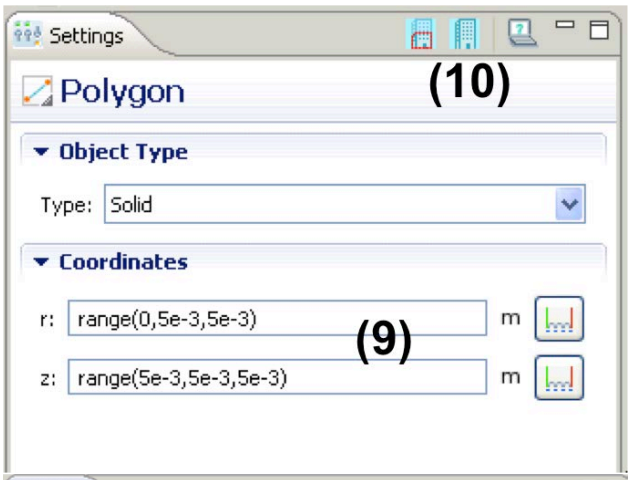
(9) Enter the following expressions in the edit fields:

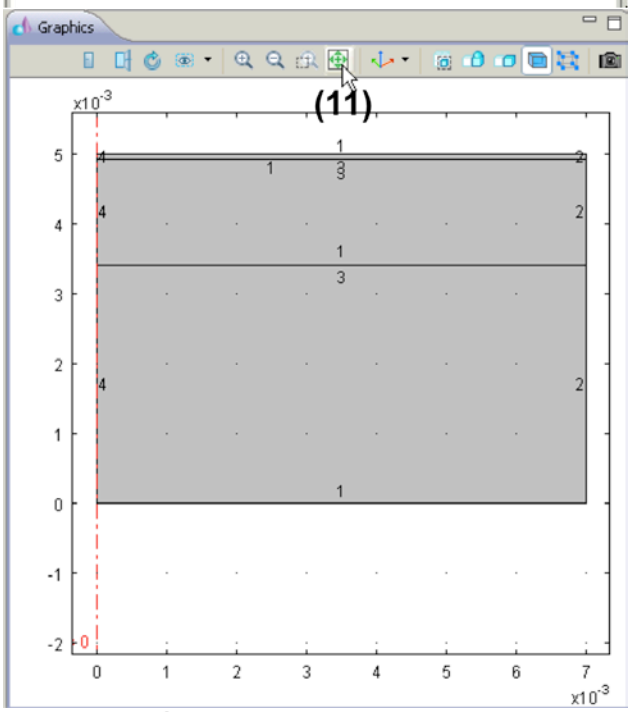
r	range(0,5e-3,5e-3)
z	range(5e-3,5e-3,5e-3)

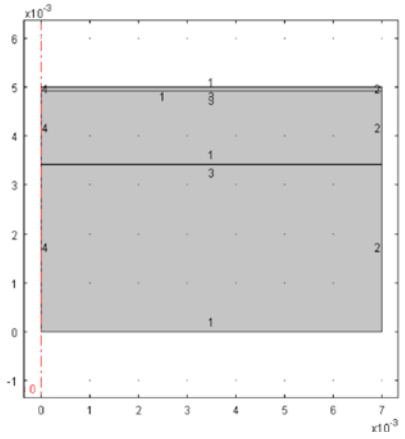
(10) Click on Build All icon.

(11) Click on Zoom Extents to fit the geometry into the window.







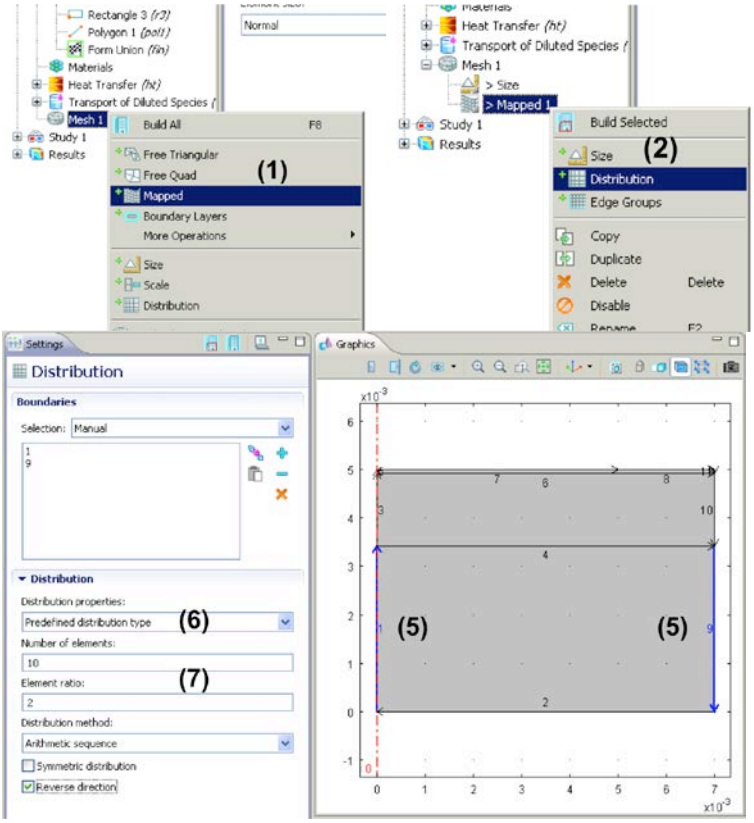


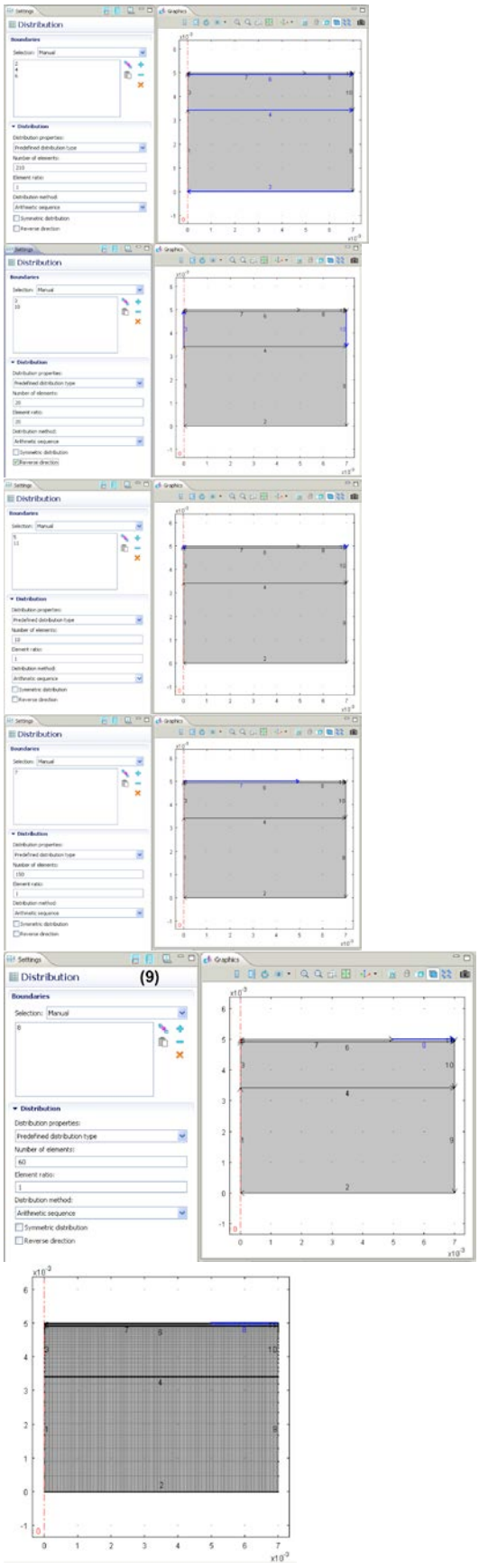
Step 3: Meshing

(1) Right click on Mesh 1 and select Mapped
(2) Right click on Mesh 1 > Mapped 1 and select Distribution
(3) Repeat (2) five more times.
(4) Click on Mesh 1 > Mapped 1 > Distribution 1
(5) Left click then right click on boundaries 1, 9.
(6) Select Predefined distribution type under Distribution properties.
(7) Set Number of elements to 10, Element ratio to 2, and check Reverse direction.
(8) Repeat (4)-(7) for Distribution 2-6.

Distribution #	Boundaries	Number of Elements	Element ratio	Reverse direction
2	2,4,6	210	1	
3	3,10	20	20	check
4	5,11	10	1	
5	7	150	1	
6	8	60	1	

(9) Click on Build All icon.





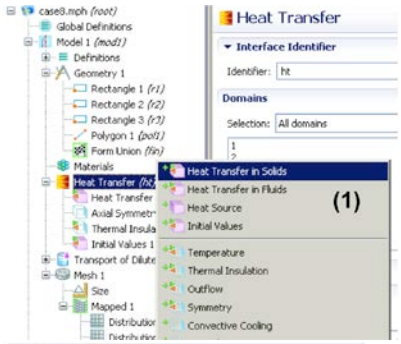
Step 4: *Conduction*—Governing Equation, Source Terms, IC, BC

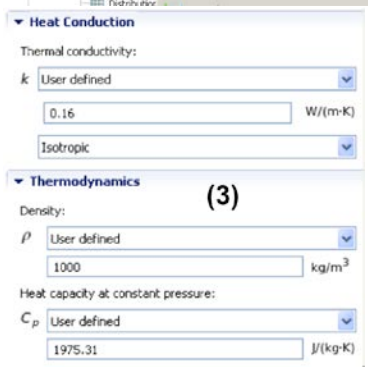
Subdomain settings

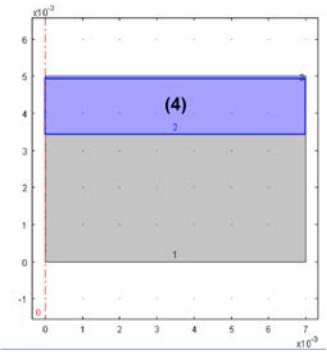
(1) Right click on Heat Transfer (ht) and select Heat Transfer in Solids
(2) Repeat (1)
(3) Click on Heat Transfer in Solids 1. Change k, ρ, Cp to User defined. Enter the information in the table below for **Subdomain 1**. Don't worry that subdomains 2&3 are also selected now; the property values for them will be re-entered in the following few steps.
(4) Click on Heat Transfer in Solids 2 and left click then right click on **subdomain 2**.
(5) Enter the information in the table below for Subdomain 2.
(6) Click on Heat Transfer in Solids 3 and left click then right click on **subdomain 3**.
(7) Enter the information in the table below for Subdomain 3.
(8) For reference, Heat Transfer in Solids 1 should now look like the following; you can see that subdomains 2&3 have been overridden.

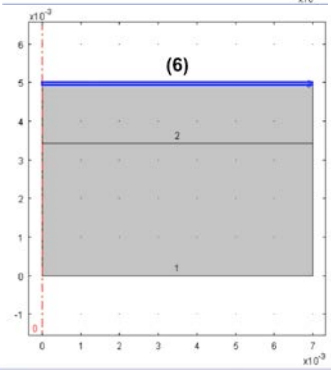
Properties

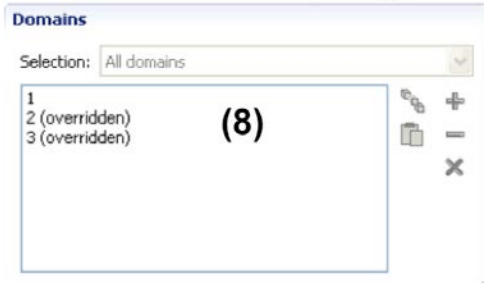
Subdomain	k	ρ	Cp
1	0.16	1000	1975.31
2	0.37	1000	2846.15
3	0.21	1000	3181.82







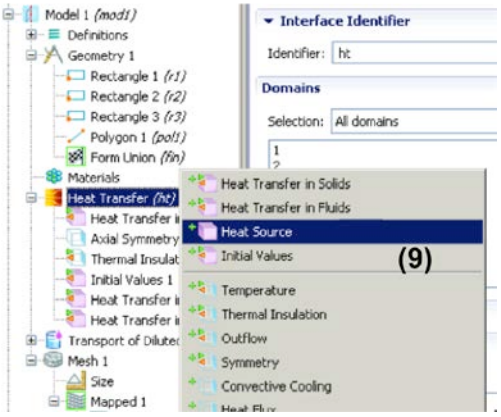


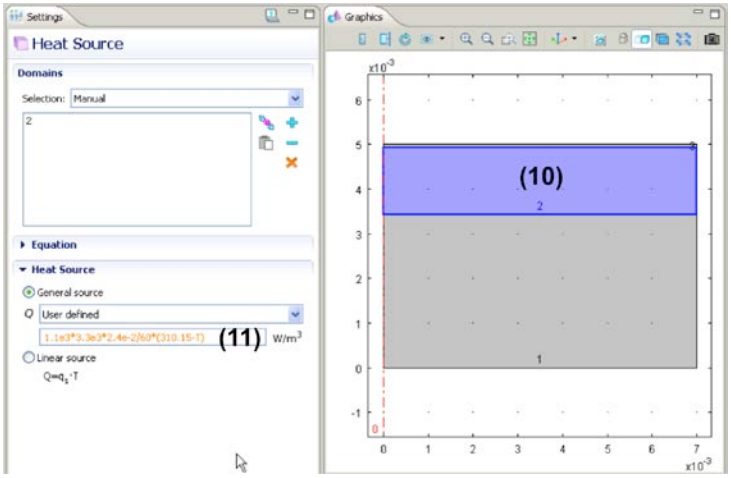


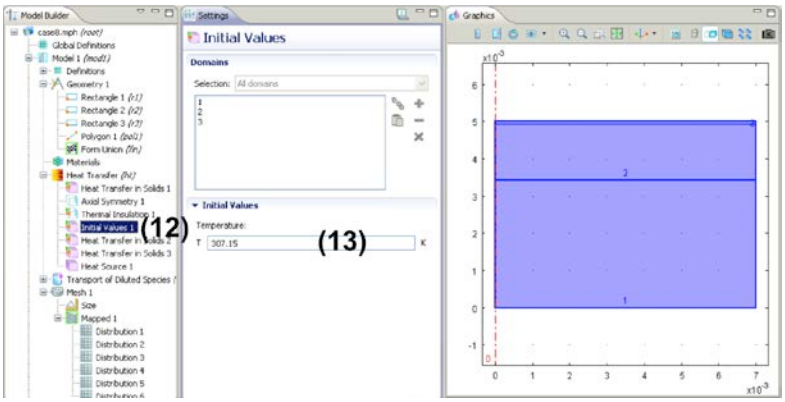
(9) Right click on Heat Transfer (ht) and select Heat Source.
(10) Left click then right click on subdomain 2.
(11) Select General Source and enter the following expression for Q:

1.1e3*3.3e3*2.4e-2/60*(310.15-T)

(12) Click on Initial Values 1
(13) Make sure all domains (1,2,3) are selected and enter 307.15 as initial temperature.

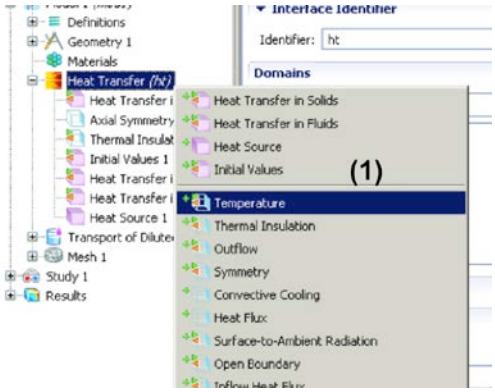


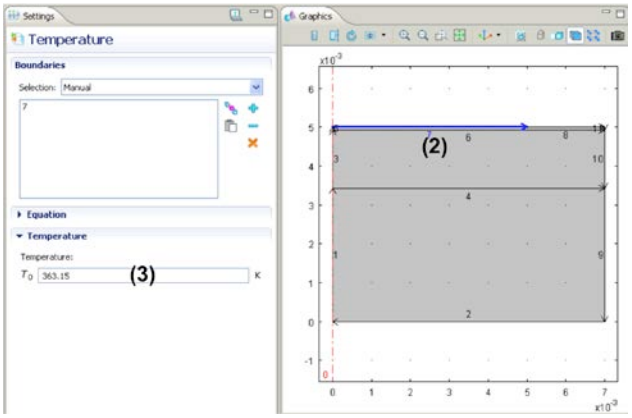




Boundary settings

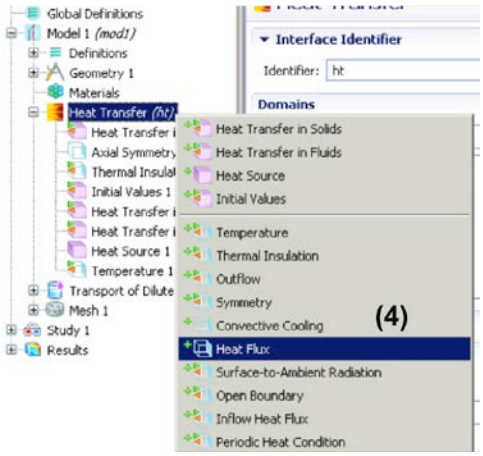
(1) Right click on Heat Transfer (ht) and select Temperature
(2) Left click and right click on boundary 7.
(3) Enter 363.15 for T_0.

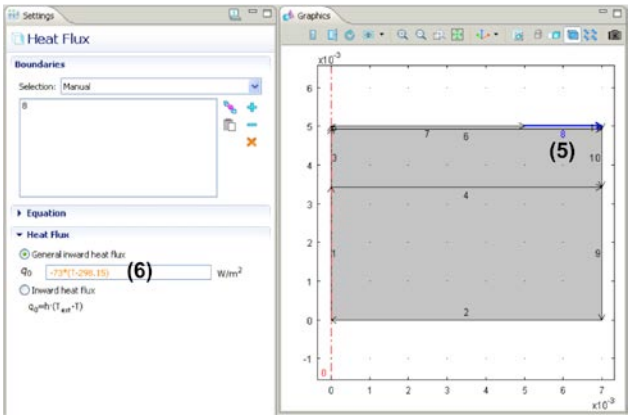




(4) Right click on Heat Transfer (ht) and select Heat Flux.
(5) Left click and right click on boundary 8.
(6) Select General inward heat flux and enter the following expression for q_0:

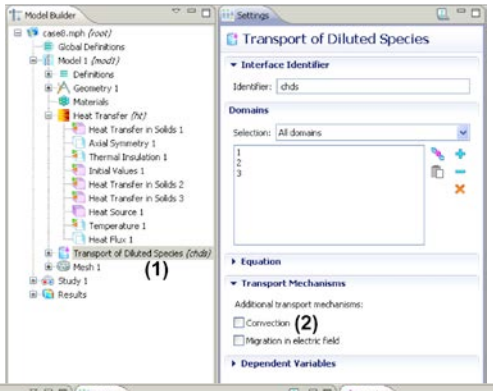
-73*(T-298.15)

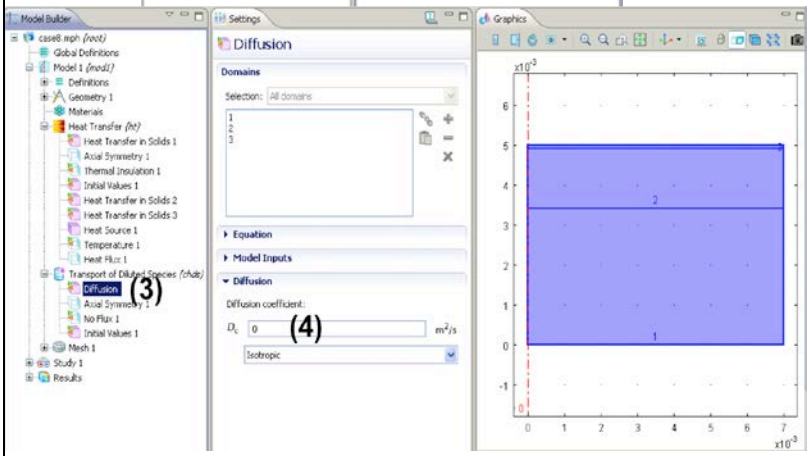




(4) Make sure all domains are selected and set Diffusion coefficient Dc to 0.
(5) Right click on Transport of Diluted Species (chds) and select Reactions.
(6) Left click and right click on all domains.
(7) Enter the following expression for Rc:

3e98*exp(-6.3e8/(8.314e3*T))

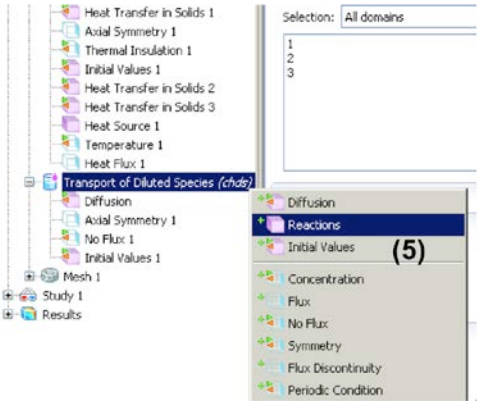


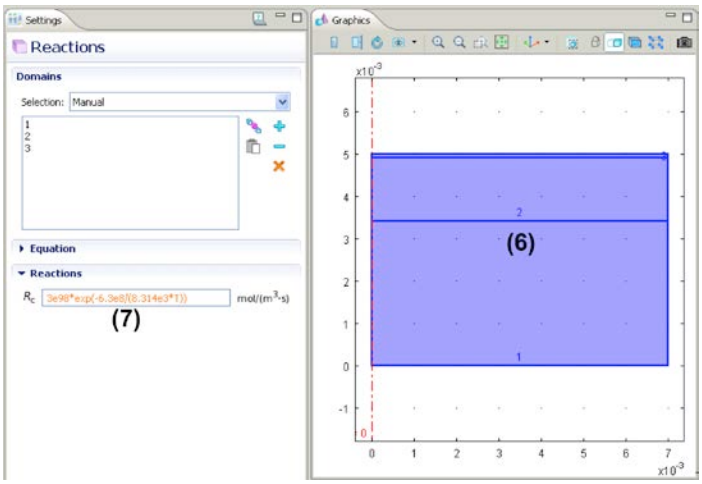


Step 5: *Diffusion*—Governing Equations, Source Terms, IC, BC

<u>Subdomain settings</u>

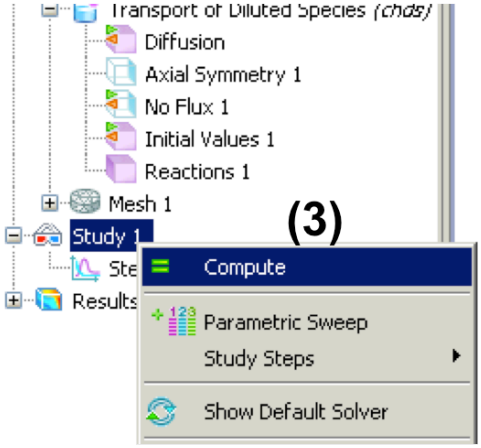
(1) Click on Transport of Diluted Species (chds)
(2) Deselect Convection
(3) Click on Transport of Diluted Species (chda) > Diffusion.





Boundary settings

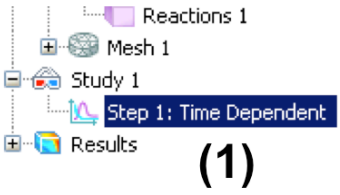
All boundaries for the species (burn) have species flux as zero due to impermeability. Since the default boundary condition of the solver is zero flux, we retain the default settings.

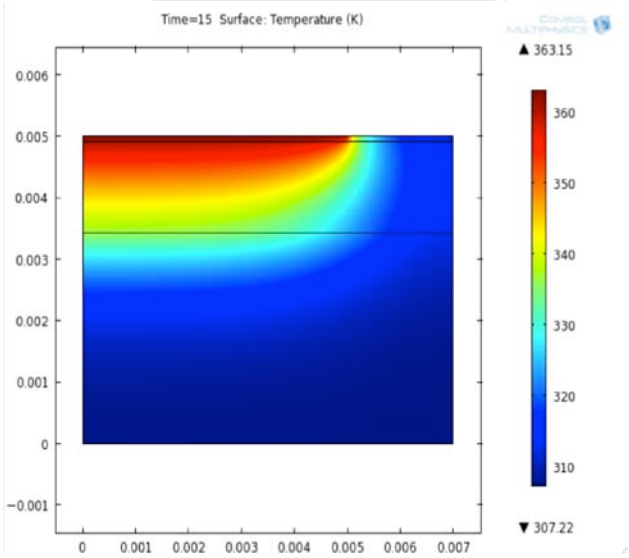


Step 6: Solver settings and Solution

(1) Select Study 1 > Step 1: Time Dependent.
(2) Type range (0,1,15) in the Times edit field.
(3) Right click on Study 1 and select Compute.

Note that the computation for this case study will take about 2.5-3 hours.

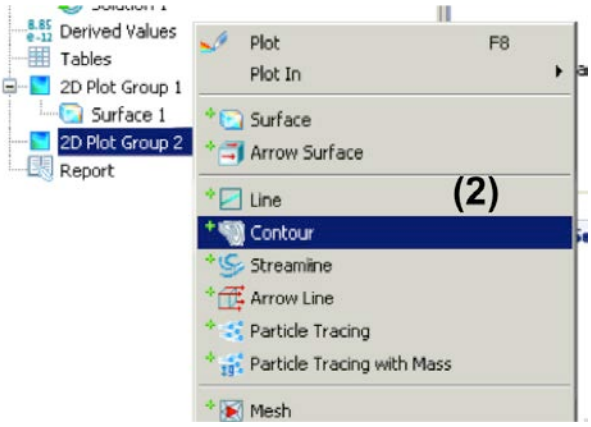


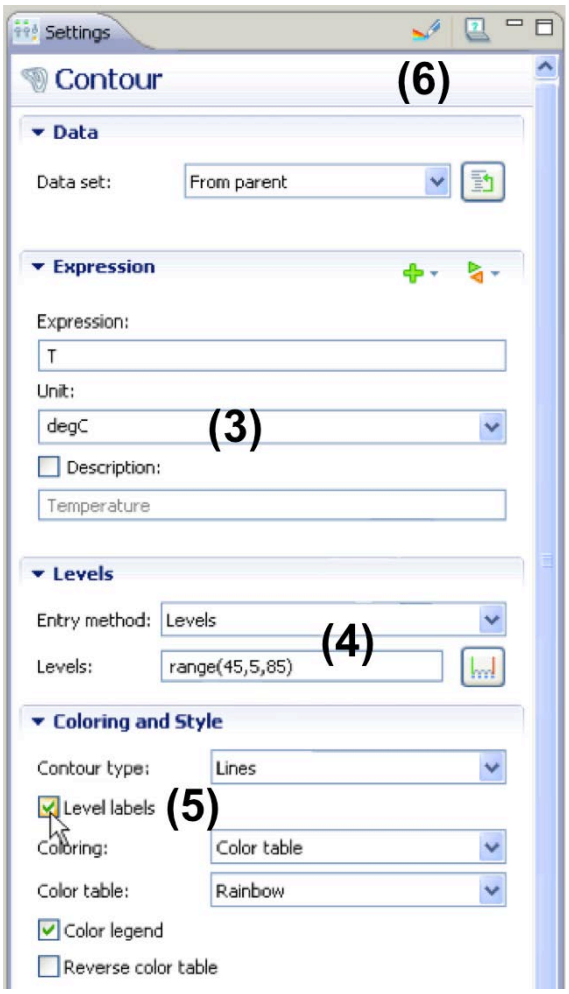


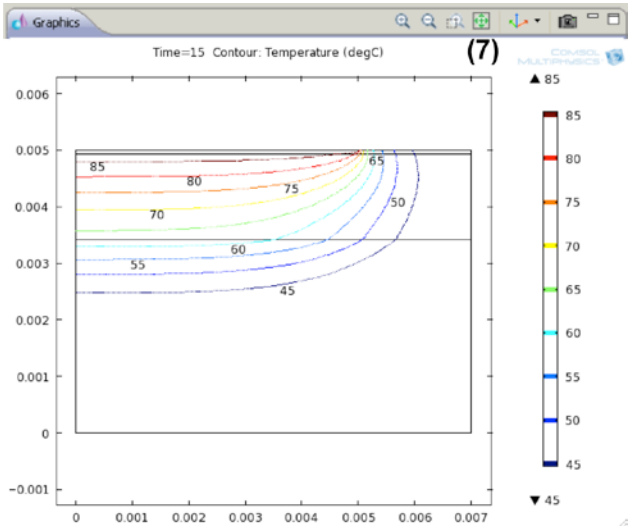
Step 7: Postprocessing

We will plot the temperature contours levels of 85, 80, 75, 70, 65, 60, 55, 50, and 45C in the domain at the end time, as shown in Diller and Hayes (1983). To generate this plot:

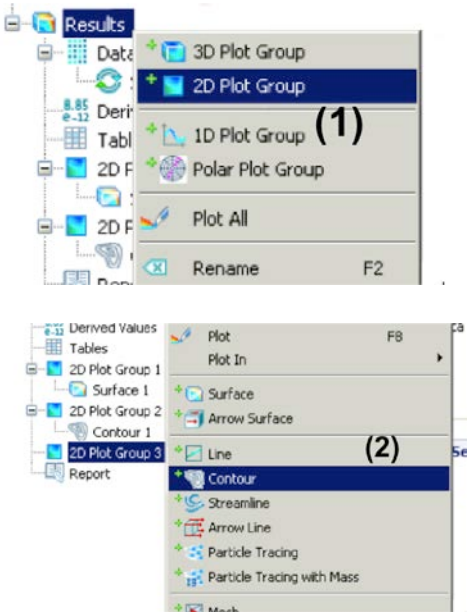
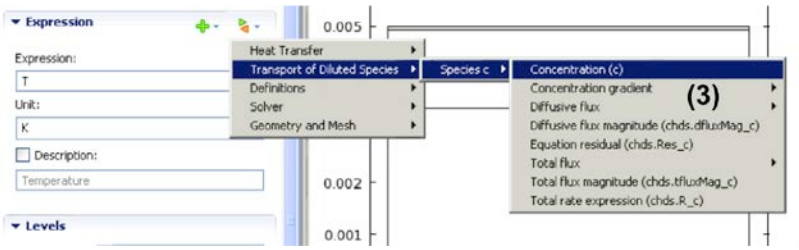
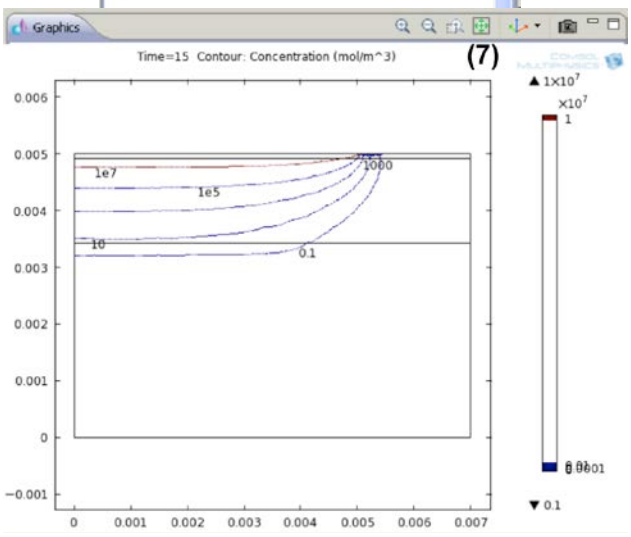
(1) Right click on Results and select 2D Plot Group.
(2) Right click on 2D Plot Group 2 and select Contour.
(3) Change the unit of temperature to degC.
(4) Change Entry methods to Levels and type range(45,5,85) in the Levels edit field.
(5) Check Level labels under Coloring and Style tab.
(6) Click on Plot icon.
(7) Click on Zoom Extents.







We will now plot the burn injury contour levels of 1e7, 1e5, 1e3, 10, and 0.1 in the domain at the end time, as shown in Diller and Hayes (1983). To generate this plot:

(1) Right click on Results and select 2D Plot Group.
(2) Right click on 2D Plot Group 3 and select Contour.
(3) Click on the double arrow next to the Expression tab title and select Transport of Diluted Species > Species c > Concentration (c).
(4) Change Entry methods to Levels and type 1e-1 1e1 1e3 1e5 1e7 in the Levels edit field.
(5) Check Level labels.
(6) Click on Plot icon.
(7) Click on Zoom Extents.

Values of burn injury (Ω) of 0.53, 1, and 10^4 correspond to injury threshold, second, and third degree burn, respectively (Diller and Hayes, 1983). The figure shows the level of burns inside the tissue after the skin is heated by the disc for 15s.