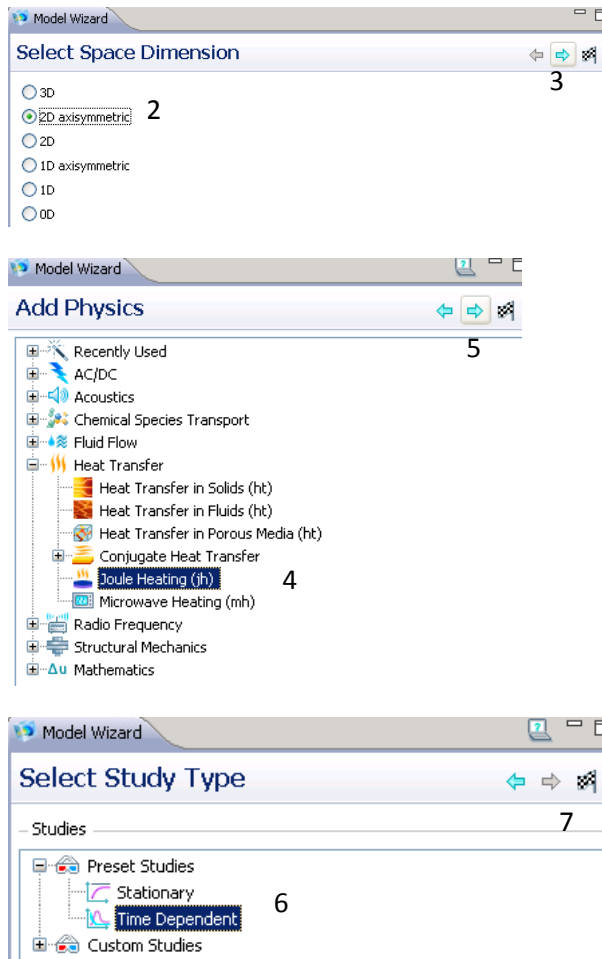
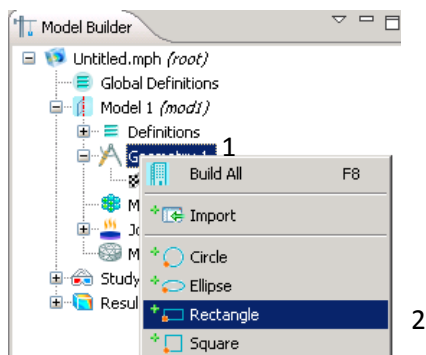


Problem Type Specification (Electro-thermal interaction module)

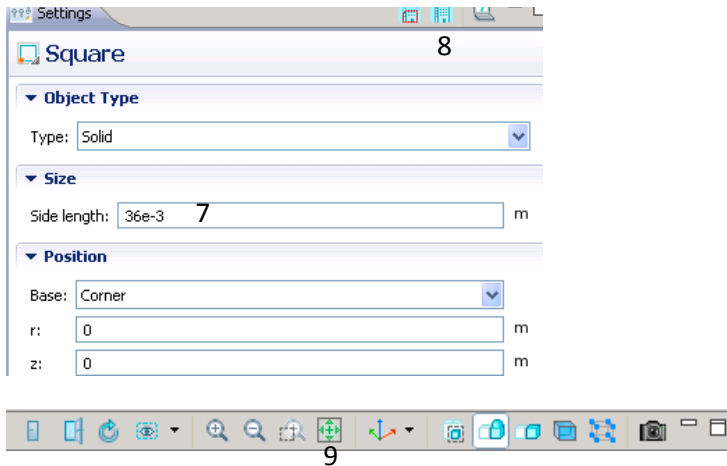
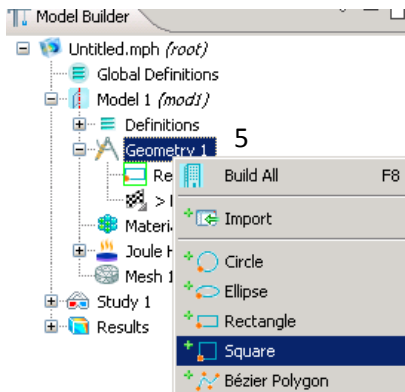
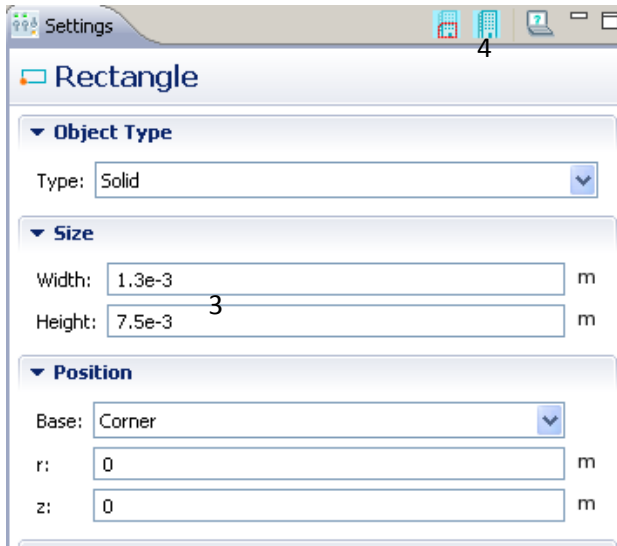


- 1) Open COMSOL 4.1 and save file. Remember to save often to prevent loss of work.
- 2) Under **Model Wizard** select "2D axisymmetric".
- 3) Click on the blue next icon.
- 4) Expand "Heat Transfer">> "Joule Heating (jh)".
- 5) Click on the blue next icon.
- 6) Under **Select Study Type**, click on "Time Dependent".
- 7) Click on checked flag icon to finish.

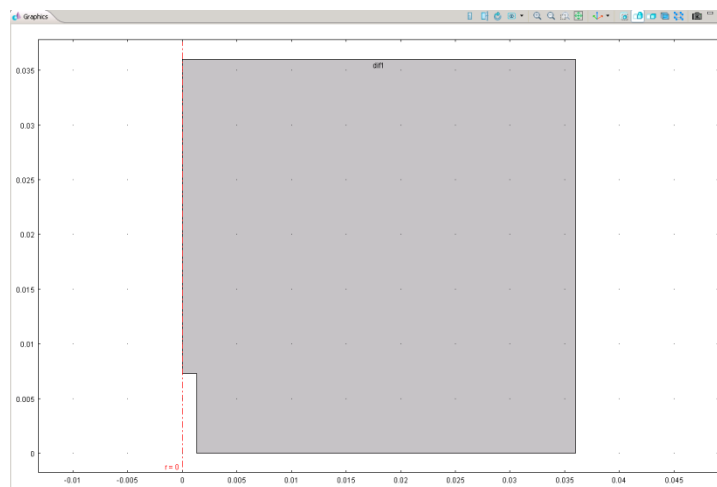
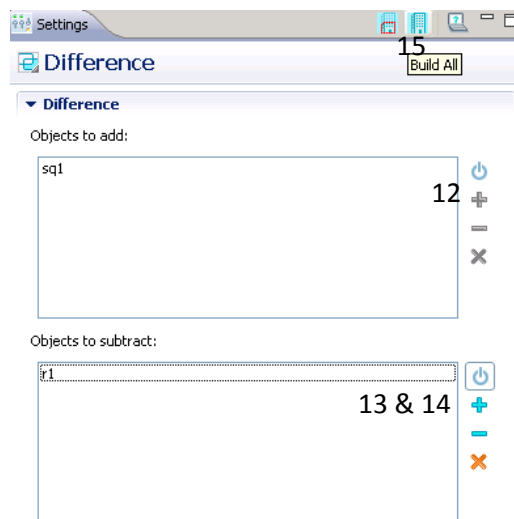
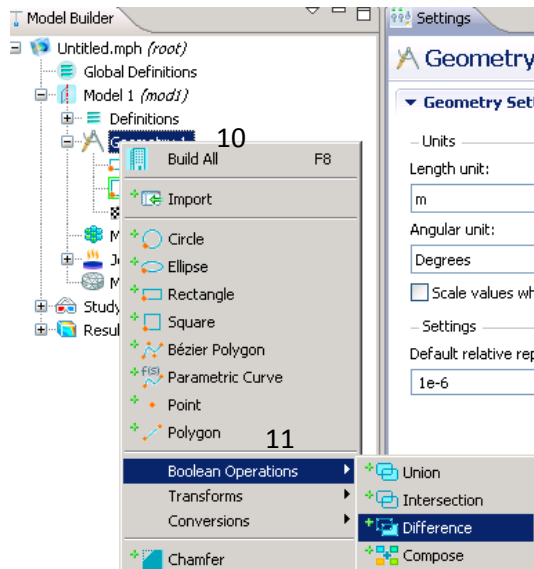
Geometry Creation



- 1) Right click on "Geometry 1".
- 2) Select "Rectangle".

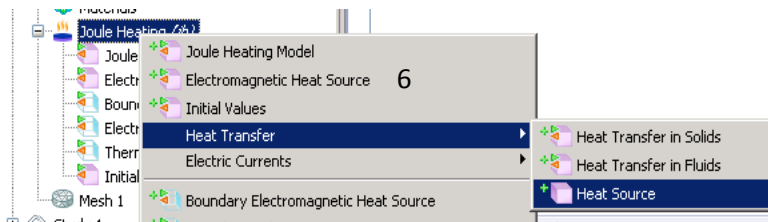
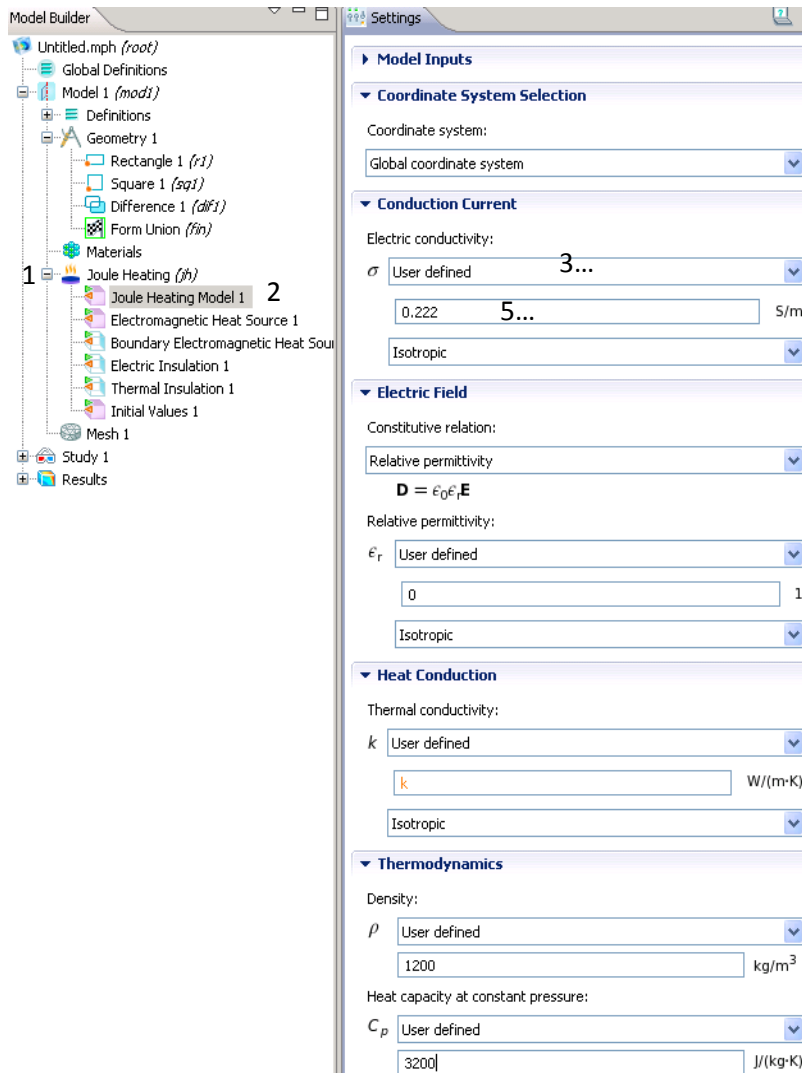


- 3) Under **Rectangle** input width of $1.3e-3$ m and height of $7.5e-3$ m. This will represent the electrode.
- 4) Click on blue build all icon
- 5) Right click on "Geometry 1".
- 6) Select "Square".
- 7) Under **Square**, input $36e-3$ m for side length.
- 8) Click on blue build all icon.
- 9) Under **Graphics**, click on green expand all icon. This will fit build geometry to window.



- 10) Right click on “Geometry 1”.
- 11) “Boolean Operations”>> Select “Difference”.
- 12) Under **Graphics**, click on the square subdomain sq1 and press the blue plus icon next to window for “Objects to add:”
- 13) Click the window for “Objects to subtract:”
- 14) Click on rectangle subdomain r1 and press the blue plus icon next to window for “Objects to subtract:”
- 15) Click the blue build all icon. This will subtract the smaller rectangle from the larger square subdomain, to leave one domain that represents the tissue surrounding the electrode to be modeled.

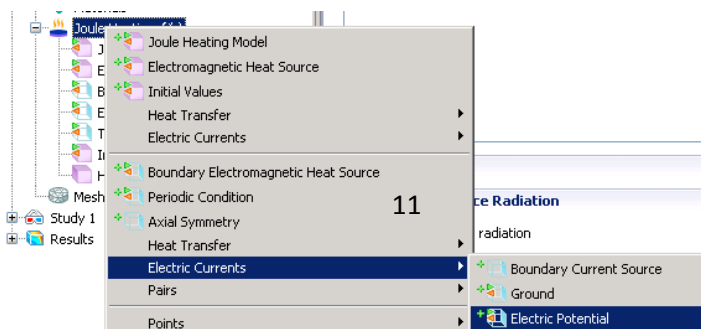
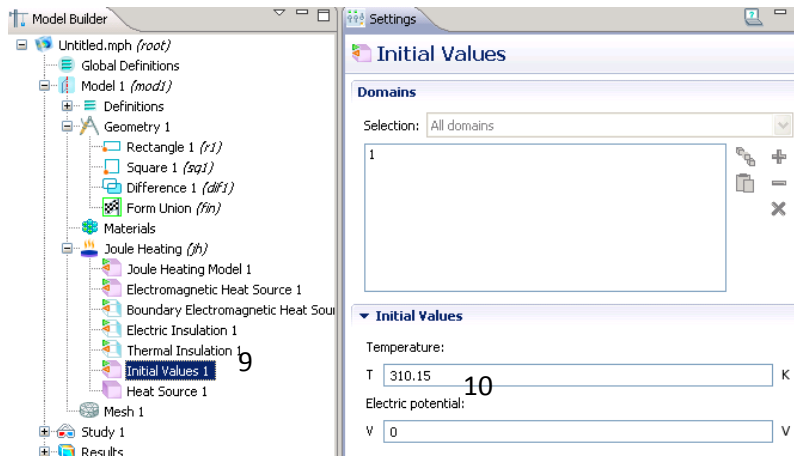
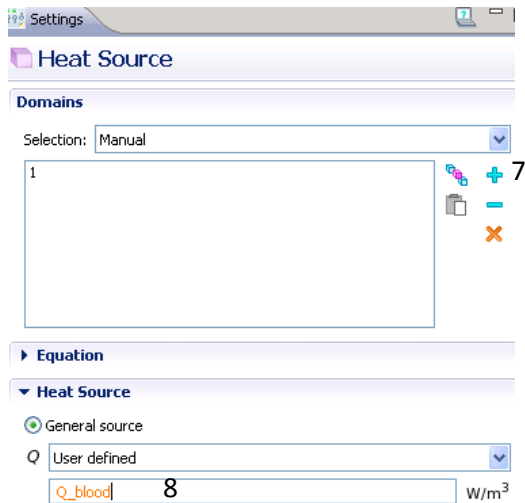
Subdomain and Boudary Settings



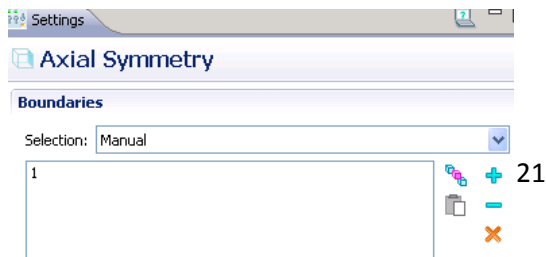
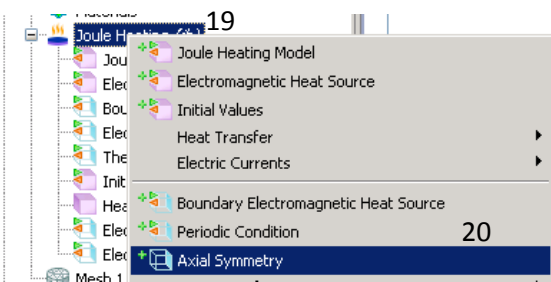
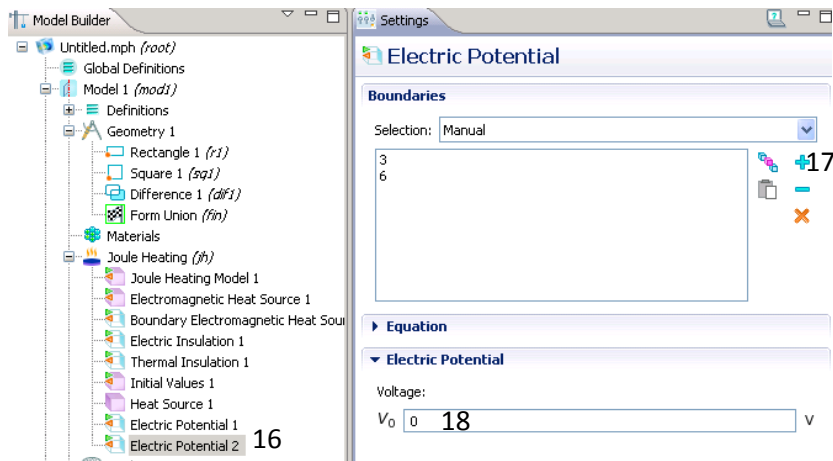
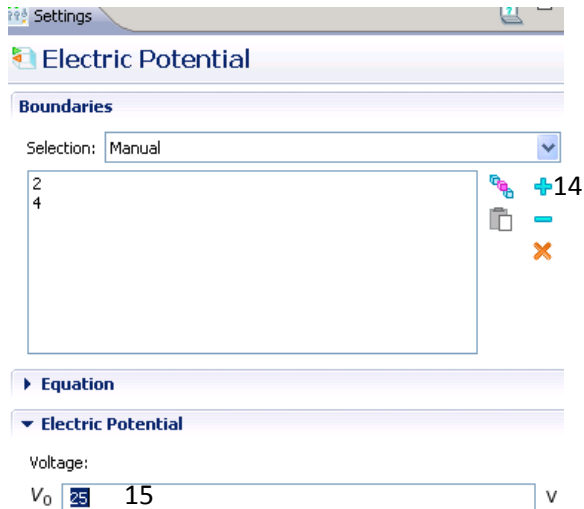
- 1) Expand “Joule Heating (jh)”.
- 2) Select “Joule Heating Model 1”.
- 3) Under **Settings**, select “user defined” in the drop down menu for electric conductivity.
- 4) Repeat for all other properties: Relative permittivity, Thermal conductivity, Density, and Heat capacity.

Property	value
Σ	0.222
ϵ_r	0
K	k
P	1200
c_p	3200

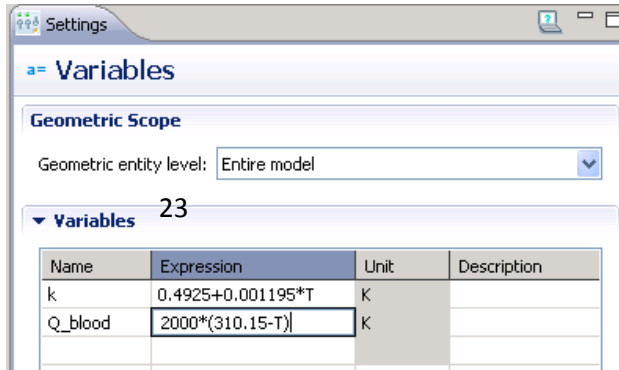
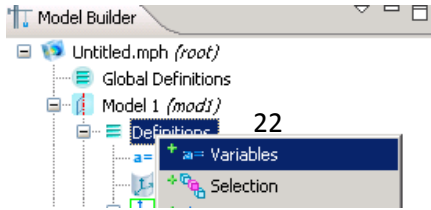
- 5) Input values for σ , ϵ_r , k, ρ , and c_p . The expression k will be defined later.
- 6) Right Click on “Joule Heating (jh)” >> “Heat Transfer” >> “Heat Source”.



- 7) Under **Graphics**, click on domain 1 and click on blue + icon.
- 8) Input Q_blood for “General source”. The expression Q_blood will be defined later.
- 9) Under “Joule Heating (jh)” select “Initial Values 1”.
- 10) Input 310.15K for “Temperature” and leave “Electric potential” to be 0.
- 11) Right Click on “Joule Heating (jh)” >> “Electric Currents” in the boundary condition section >> “Electric Potential”.
- 12) Repeat steps (13)-(15) to create two “Electric Potential” boundary conditions.



- 13) Click on “Electric Potential 1”.
- 14) Under **Graphics**, hold CTRL key and click on boundaries 2 and 4. Click on the blue + icon.
- 15) Set “Voltage” to 25V.
- 16) Select “Electric Potential 2”.
- 17) Under **Graphics**, hold CTRL key and click on boundaries 3 and 6. Click on the blue + icon.
- 18) Set “Voltage” to 0V.
- 19) Right click on “Joule Heating (jh)”.
- 20) Select “Axial Symmetry”.
- 21) Select on boundary 1 and click the blue + icon.

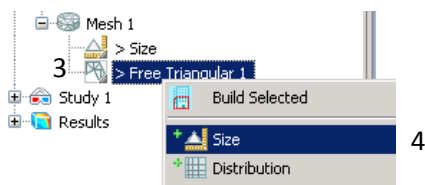
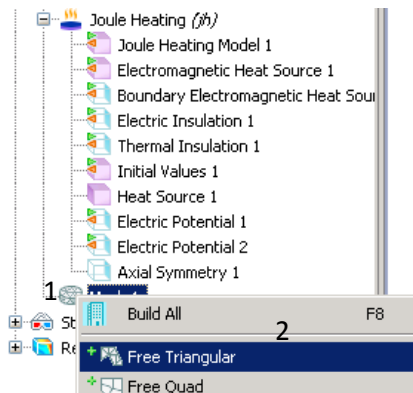


22) Right Click on "Definitions" and Select "Variables". This will allow us to define expressions that were used.

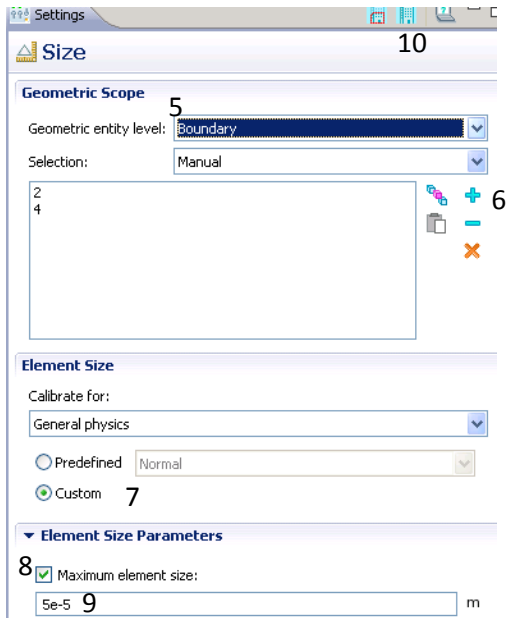
Name	Expression
k	0.4925+0.001195*T
Q_blood	2000*(310.15-T)

23) Input the above expressions in "Variables".

Meshing

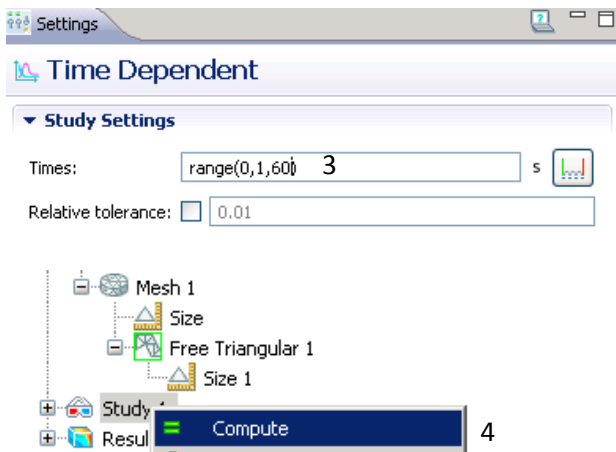
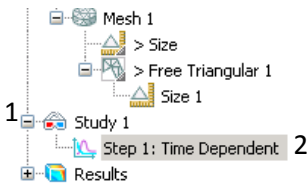


- 1) Right Click on "Mesh 1".
- 2) Select "Free Triangular".
- 3) Right click on "Free Triangular 1".
- 4) Select "Size".

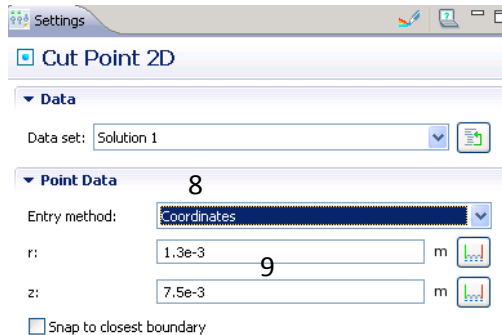
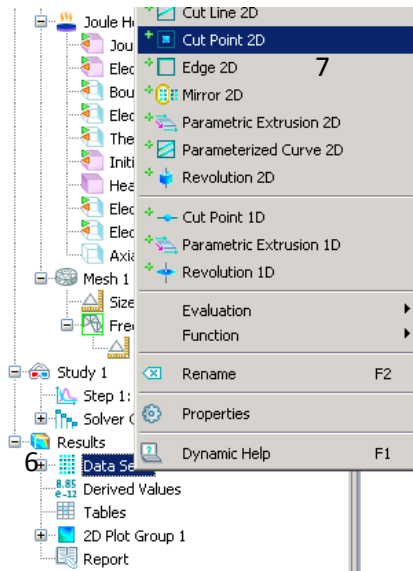
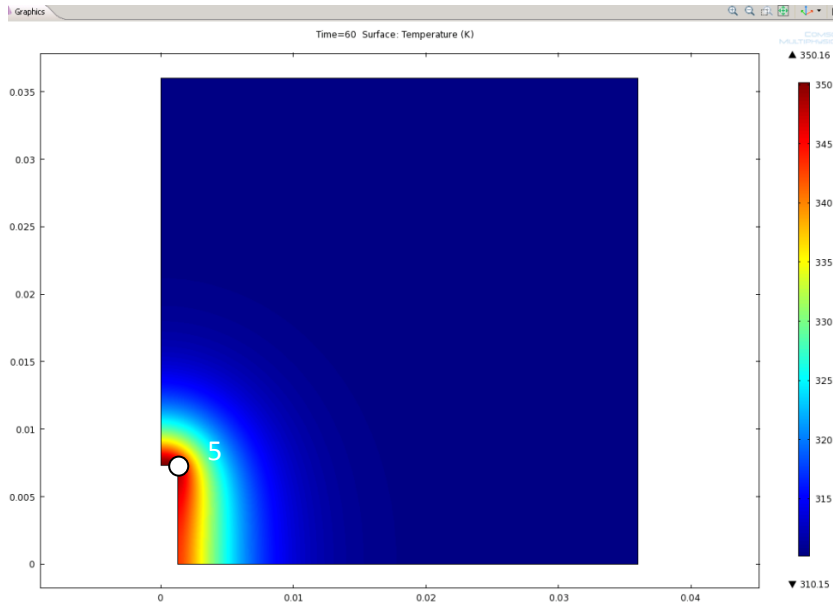


- 5) Choose “Boundary” in Geometric entry level.
- 6) Hold CTRL and select boundaries 2&4 and click on blue + icon.
- 7) Under “Element Size”, choose ‘Custom’.
- 8) Check off on “Maximum element size”.
- 9) Input 5e-5 m for maximum element size.
- 10) Click on blue build all icon.

Computation and Postprocessing

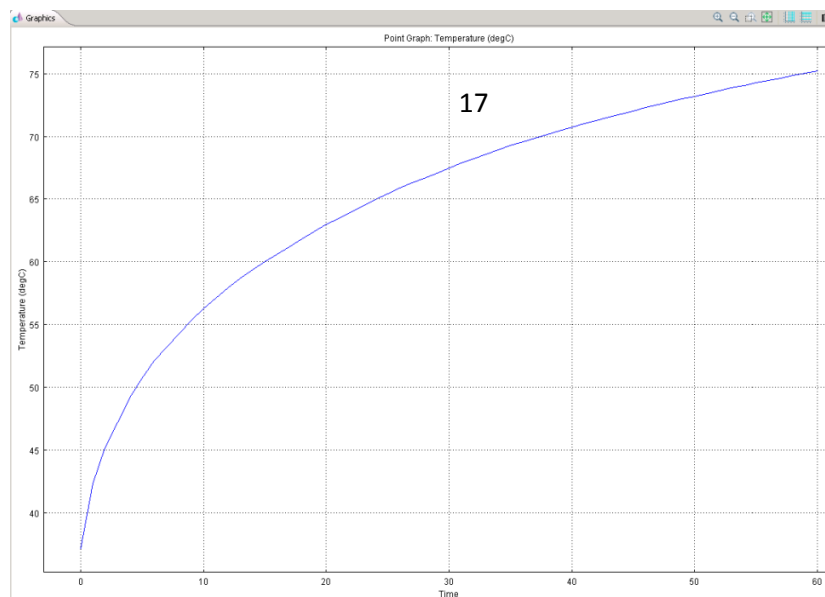
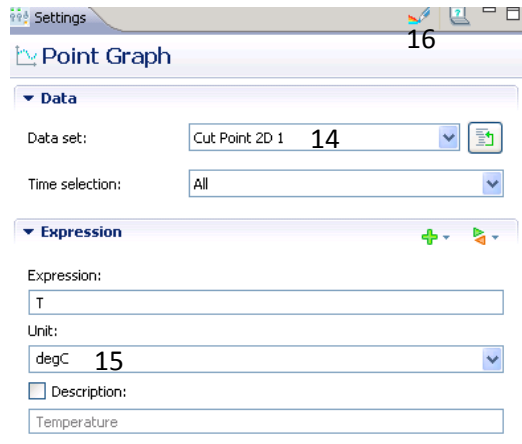
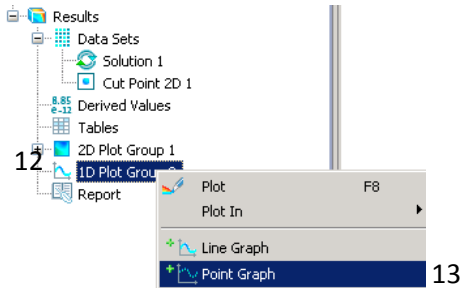
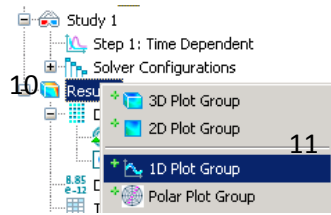


- 1) Expand “Study 1”.
- 2) Click on “Step 1: Time Dependent”.
- 3) Under **Settings**, set time range to be range(0,1,60).
- 4) Right click on “Study 1” and select “Compute”.



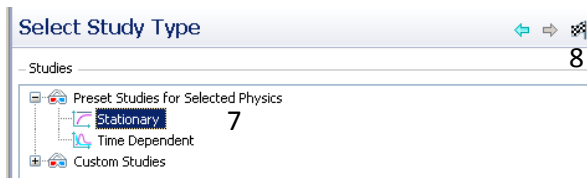
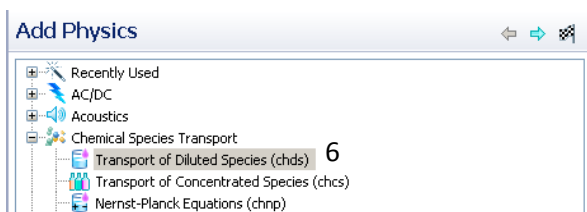
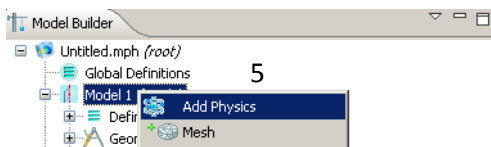
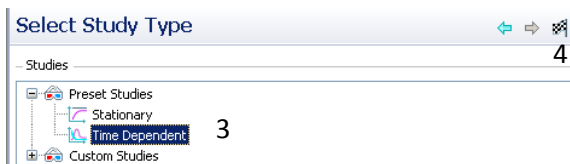
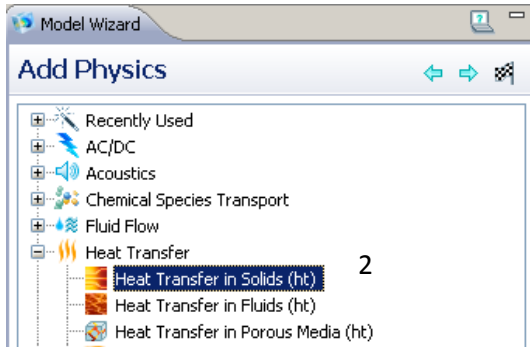
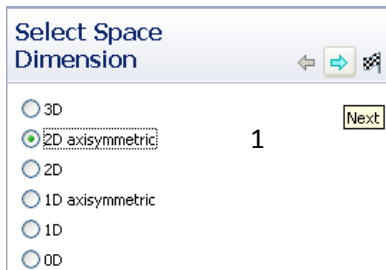
- 5) Following surface plot should result. For post-processing we want to create a temperature profile at the tip of the electrode marked by the white dot.
- 6) Right click on "Data Set".
- 7) Select "Cut Point 2D".
- 8) Under **Settings**, choose "coordinates" from the drop down menu for Entry Method.
- 9) Input polar coordinate (1.3e-3, 7.5e-3).

Note: you can click on the plot icon now and it will show where the coordinate you just inputed is on the geometry



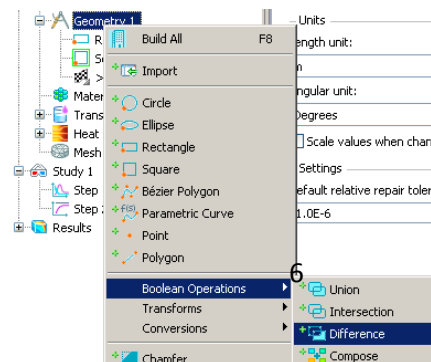
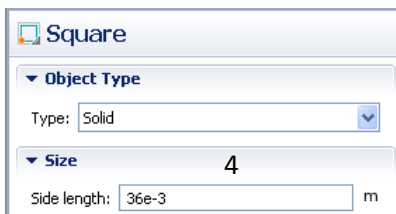
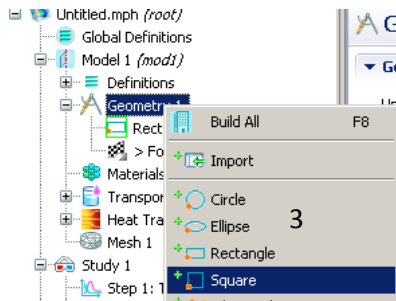
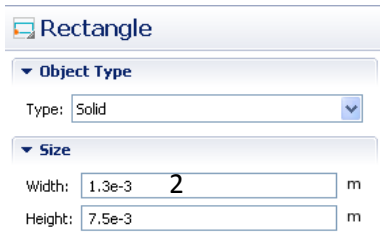
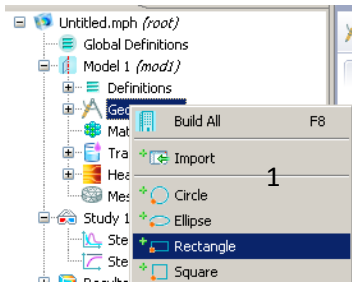
- 10) Right Click on "Results".
- 11) Select "1D Plot Group".
- 12) Right click on "1D Plot Group 2".
- 13) Select "Point Graph".
- 14) Under **Settings**, from the drop down menu for Data set, choose "Cut Point 2D 1".
- 15) Change the unit from K to degC using the drop down menu for Unit under **Expression**.
- 16) Click the rainbow plot icon.
- 17) The following temperature profile should result.

Problem Type Specification (Alternative Method)

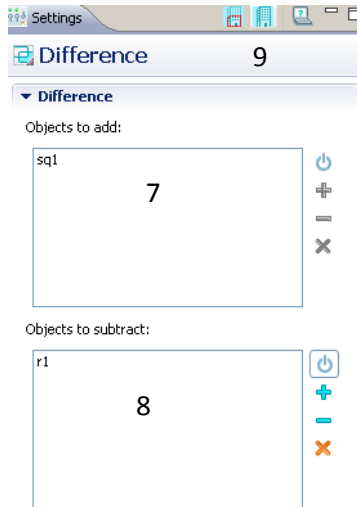


- 1) Select 2D axisymmetry.
- 2) Expand “Heat Transfer” >> “Heat Transfer in Fluids (ht)”.
- 3) Select “Time Dependent” for transient analysis.
- 4) Click checked flag to finish add desired physics.
- 5) Right click on “Model 1” and select “Add Physics”.
- 6) Under **Add Physics**, expand “Chemical Species Transport”>> “Transport of Diluted Species (chds)”.
- 7) Select “Stationary” for steady state analysis. The steady diffusion equation will be used to solve the Laplace equation for electric potential.
- 8) Click on the checked flag “Finish” icon.

Geometry

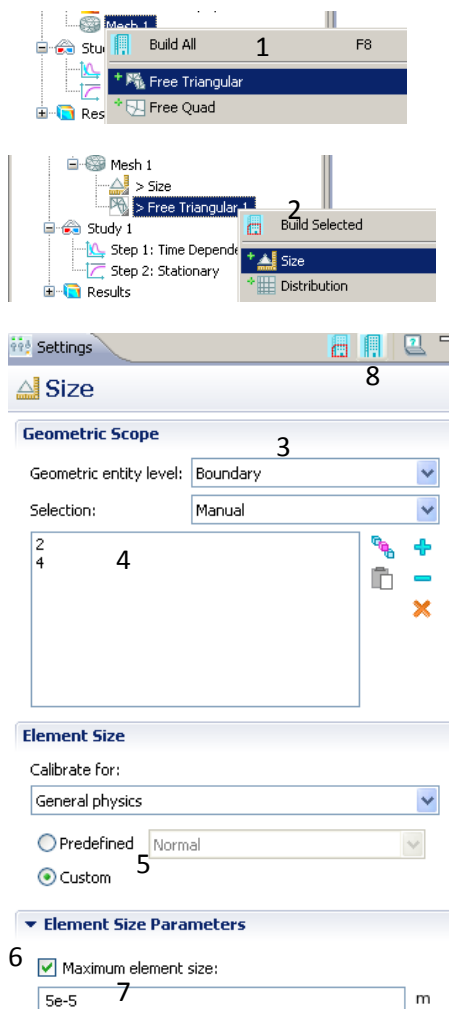


- 1) Right click on "Geometry" and select to add "Rectangle".
- 2) Input under **Rectangle**, width of $1.3e-3$ m and height of $7.5e-3$ m.
- 3) Right click on "Geometry" and select to add "Square".
- 4) Input under **Square**, side length of $36e-3$ m. Click blue "Build All" icon.
- 5) In the **Graphics** window, click on green zoom extents icon to fit geometry to window.
- 6) Right click on "Geometry">> "Boolean Operation" >> "Difference". This will be used to create a composite object.



- 7) Select the square as “Objects to add”.
- 8) Select the rectangle as “Objects to subtract”.
- 9) Click on the blue build all icon to create a composite object used as the computational domain.

Meshing



- 1) Right click on “Mesh 1” and select to add “Free Triangular” meshing.
- 2) Right click on “Free Triangular” and select to add “size”.
- 3) Select “Boundary” from the pull down menu for “Geometric entity level”.
- 4) Hold CTRL and select boundaries 2 and 4. Click the blue + to add boundaries.
- 5) Under “Element Size,” click on the “Custom” bubble.
- 6) Under “Element Size Parameters,” check off box for “Maximum element size”.
- 7) Input 5e-5 m.
- 8) Click “Build All” icon.

Properties and Parameters—Conduction

1

2

3

4

- 1) Expand “Heat Transfer (ht)” >> “Heat Transfer in Solids 1”.
- 2) Enter values for k , ρ , and c_p . The expression of k will be defined later.

Property	Value
k	K
ρ	1200
c_p	3200

- 3) Select “Initial Values 1” and input under Temperature 310.15 K.
- 4) Right click on “Heat Transfer (ht)” and add “Heat Source” term.
- 5) Add domain 1 and input Q for general heat source term.

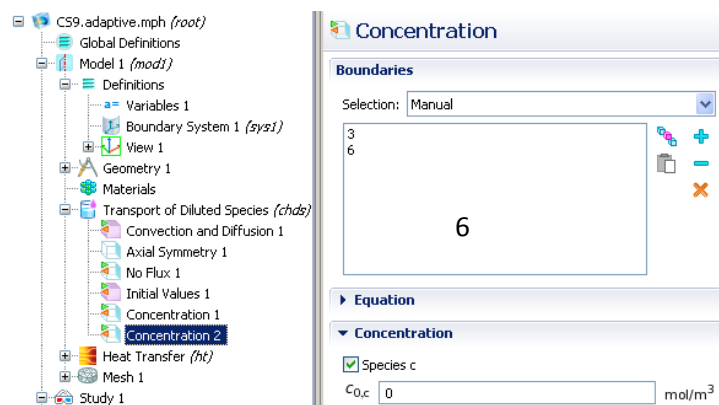
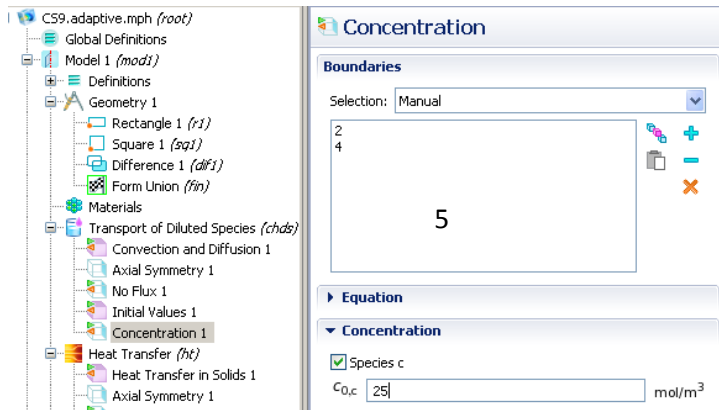
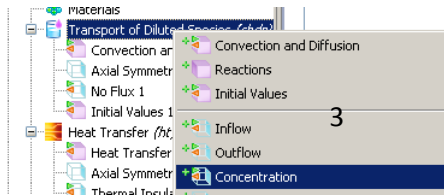
5

Properties and Parameters—Diffusion

1

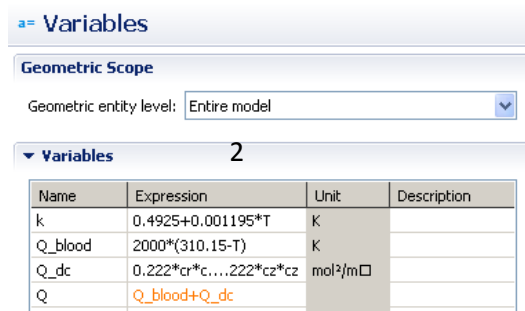
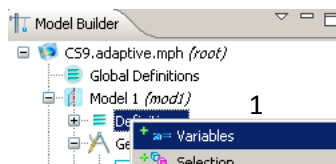
2

- 1) Expand “Transport of Diluted Species (chds)” >> “Convection and Diffusion 1”.
- 2) Under **Diffusion**, input $0.222 \text{ m}^2/\text{s}$ for the “Diffusion coefficient”.



- 3) Right click on “Transport of Diluted Species (chds)” and add “Concentration” boundary condition.
- 4) Repeat step 3 to create 2 new concentration boundary conditions.
- 5) Add boundaries 2 and 4 to “Concentration 1” and set concentration to 25.
- 6) Add boundaries 3 and 6 to “Concentration 2” and set to 0.

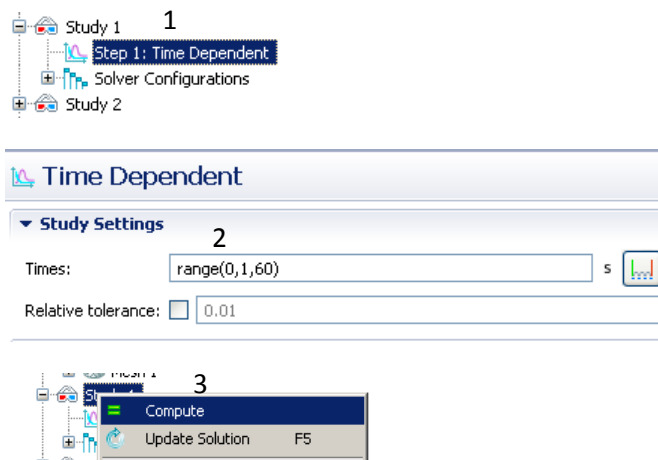
Define Expressions



- 1) Right click on “Definitions” and add new “Variables”.
- 2) Under **Variables**, input following expressions:

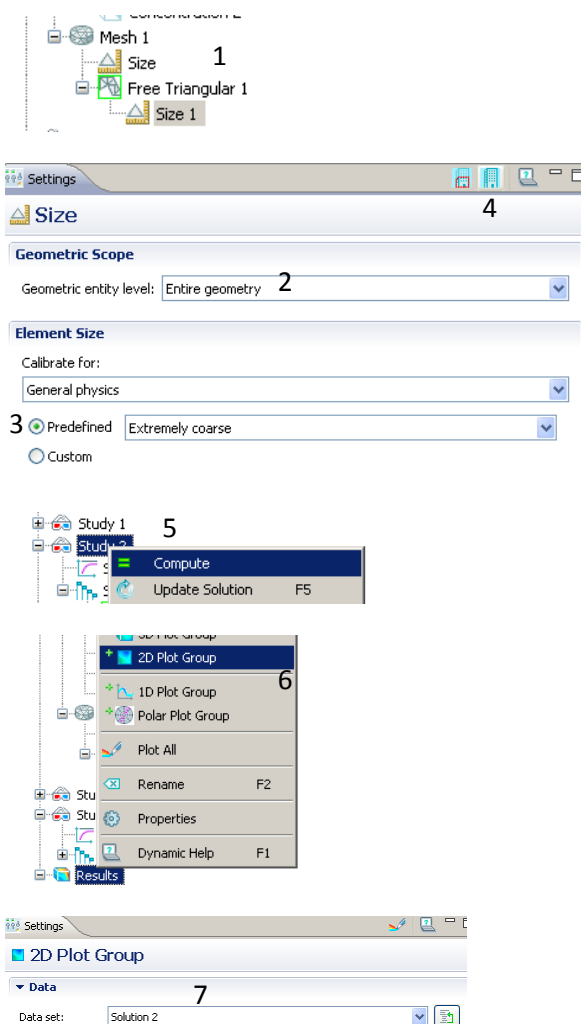
Name	Expression
k	0.4925 + 0.001195*T
Q_blood	2000*(310.15-T)
Q_dc	0.222*cr*cr + 0.222*cز*cز
Q	Q_blood + Q_dc

Solver Settings

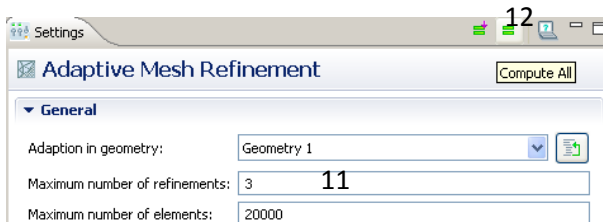
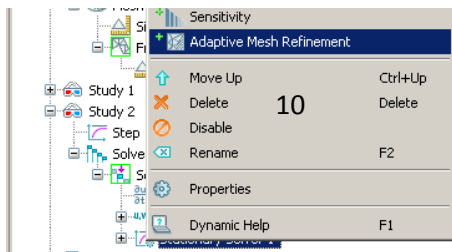
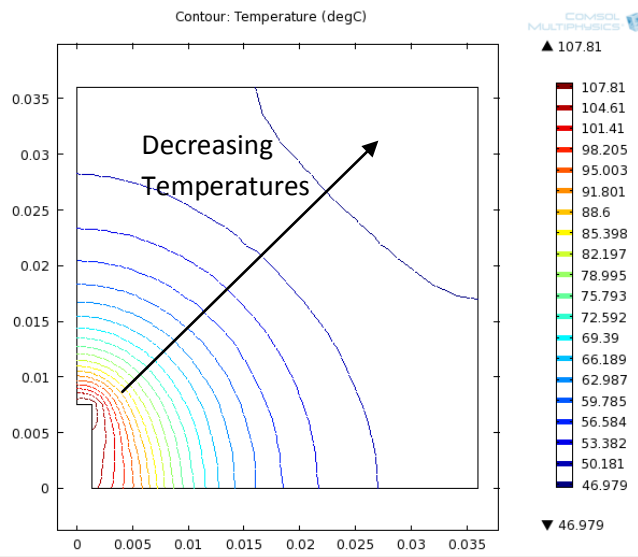
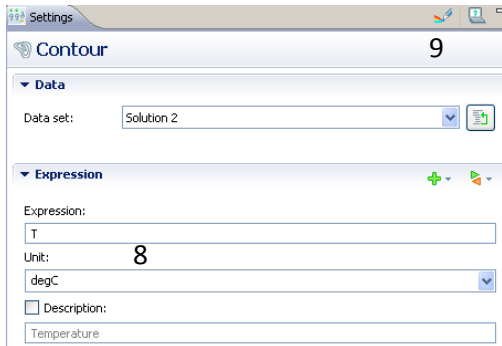


- 1) Expand “Study 1” and click on “Step 1: Time Dependent”.
- 2) Under **Time Dependent**, input range(0,1,60) in the “Times” edit field.
- 3) Right click on “Study 1” and select “Compute”.

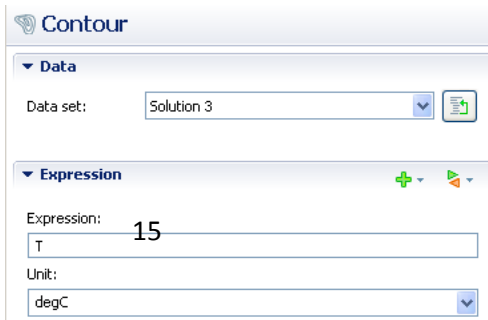
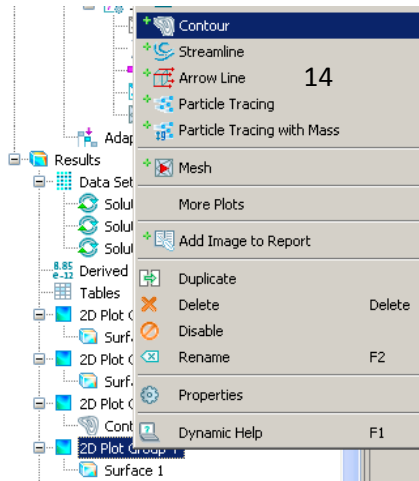
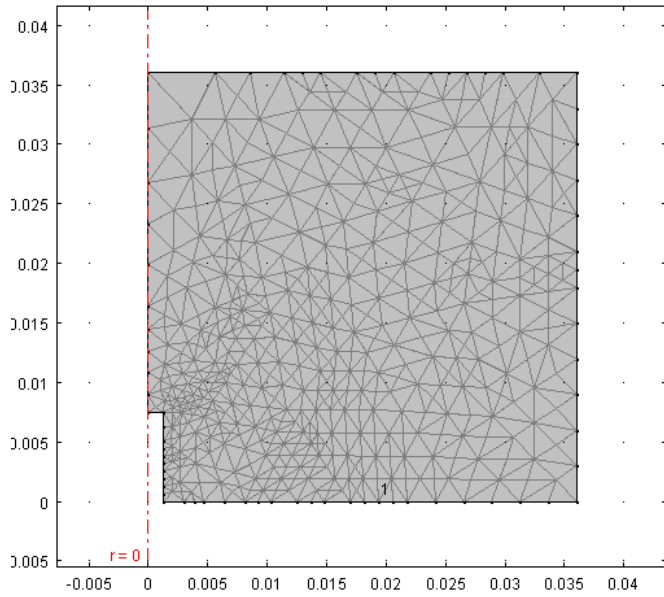
Postprocessing—Adaptive mesh refinement



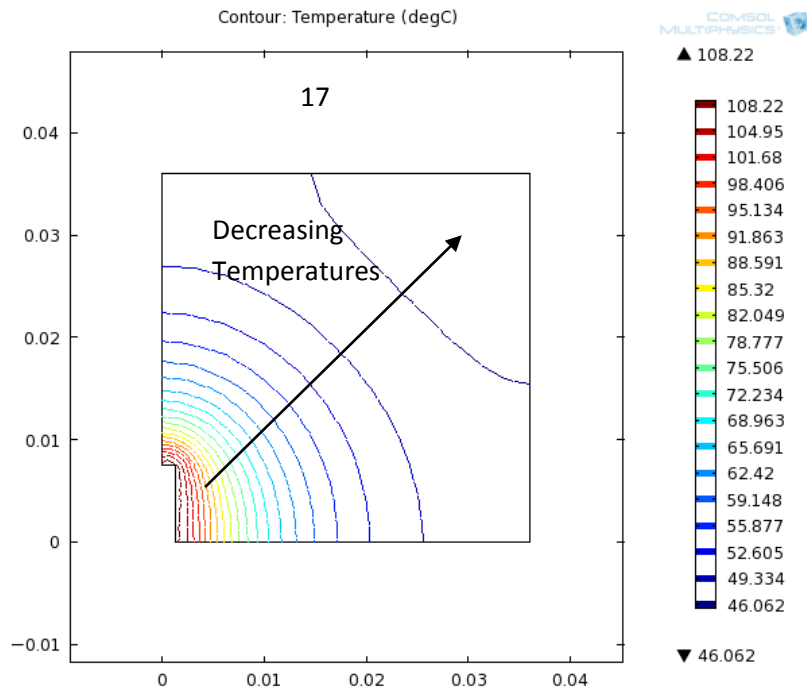
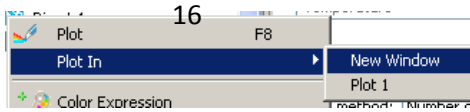
- 1) Expand “Mesh 1” >> “Free Triangular 1” >> select “Size 1”.
- 2) Under **Size**, change Geometric entity level to “Entire geometry”.
- 3) Click on bubble for “Predefined” and select “Extremely coarse” from the drop-down menu.
- 4) Click on the blue build all icon to create new mesh.
- 5) Right click on “Study 2” and select “Compute”.
- 6) Right click on “Results” and add new “2D Plot Group”.
- 7) Under **2D Plot Group**, select “Solution 2” from the pull down menu for Data set.



- 8) Right Click on “2D Plot Group 3” and add new “Contour” plot. Under **Contour**, change unit to “degC” to report temperature in degrees Celsius
- 9) Click the plot icon. The following is the resulting contour plot in degrees Celsius.
- 10) Expand “Study 2” >> “Solver Configurations” >> “Solver 2”>> right click on “Stationary Solver 1” and add new “Adaptive Mesh Refinement”.
- 11) Under **Adaptive Mesh Refinement**, set maximum number of refinements to 3 and maximum number of elements to 20,000.
- 12) Click the green “Compute All” icon to compute the final solution using adaptive mesh refinement.



- 13) Click on “Mesh 2” to view the new meshing using adaptive mesh refinement. The following is the resulting adaptive refinement mesh.
- 14) Right click on “2D Plot Group 4” and add new “Contour” plot.
- 15) Under **Contour**, select “Solution 3” for the Data set and change the unit to “degC” to produce a contour plot with degrees celsius.



16) Right click on “Contour 1” from “2D Plot Group 4” >> “Plot in” >> “New Window”.

17) The following the resulting contour plot in degrees Celsius.