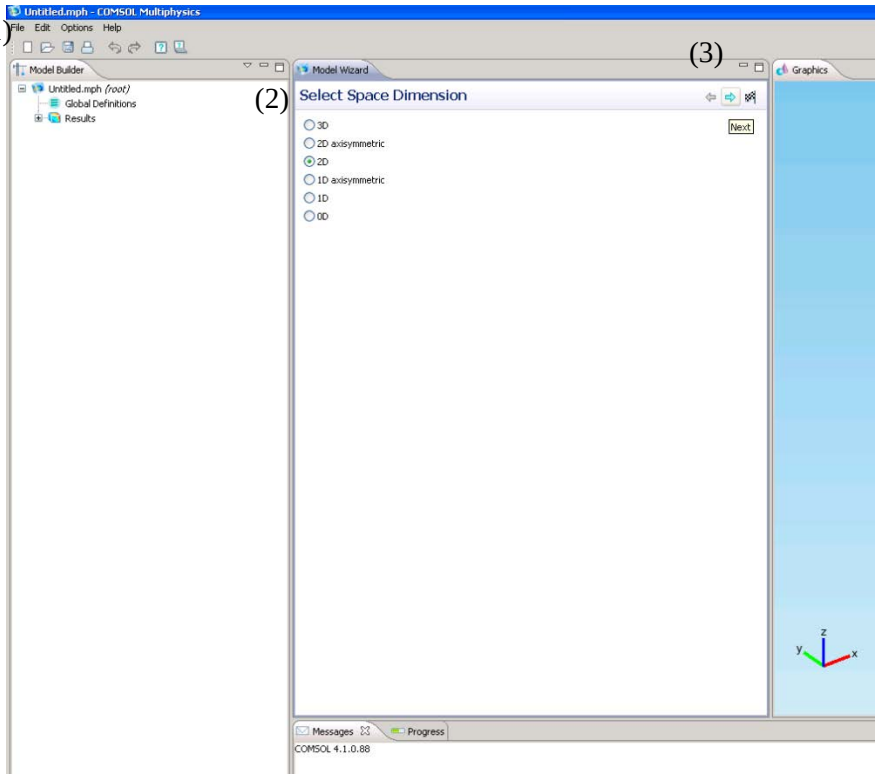
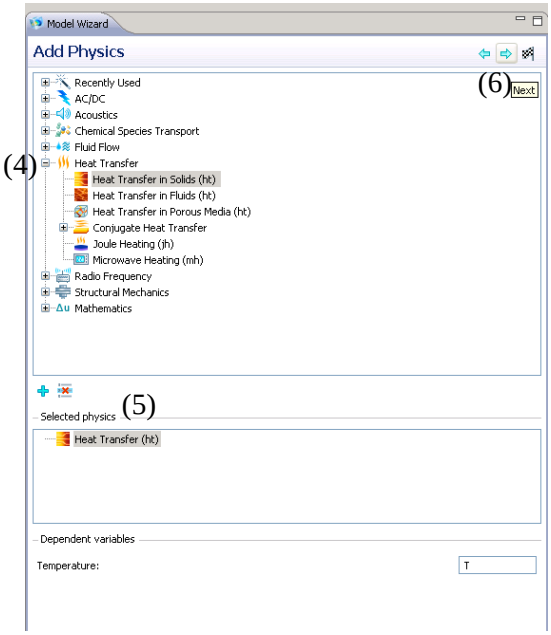
## Specifiying the Problem Type




(1) Start COMSOL 4.1 and save file. Remember to save often to prevent losing work.

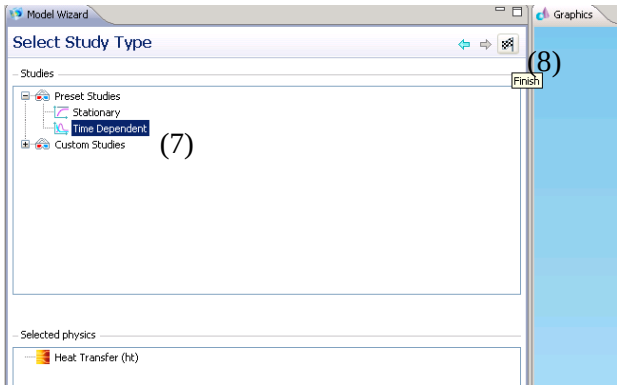
(2) Select 2D under **Select Space Dimension.**

(3) Click on blue arrow for "Next".

(4) Under **Add Physics,** click on the icon left of "Heat Transfer" to expand options >> "Heat Transfer in Solids (ht)".

(5) "Heat Transfer (ht)" should now appear in the **Selected Physics** window near the bottom of the screen.

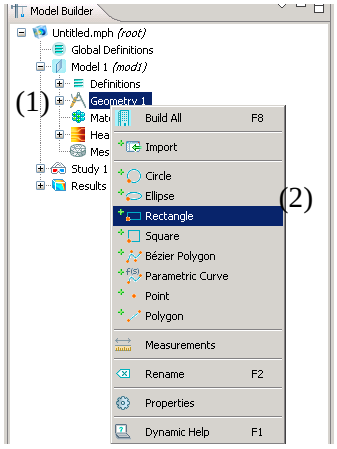
(6) Click on blue arrow Next arrow to continue.




(7) Under **Select Study Type**, select “Time Dependent” to solve as a time-dependent conduction problem.

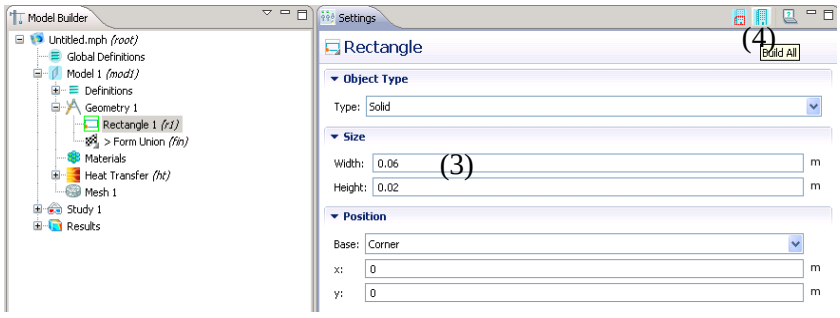
(8) Click on checkered flag to finish building desired physics.

## Creating the Geometry



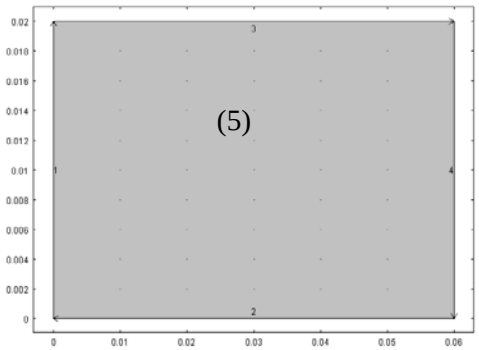

(1) Under the **Model Builder** tab, Right click on “Geometry 1”.

(2) Select to add new “Rectangle” geometry.



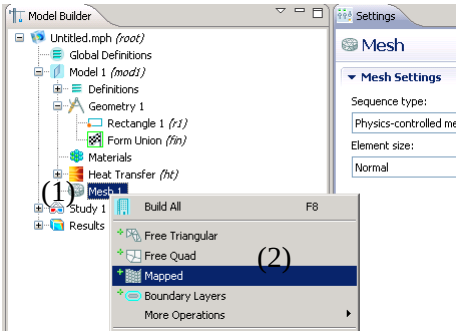

(3) In **Settings** window, set width of rectangle as 0.06m and height as 0.02m.

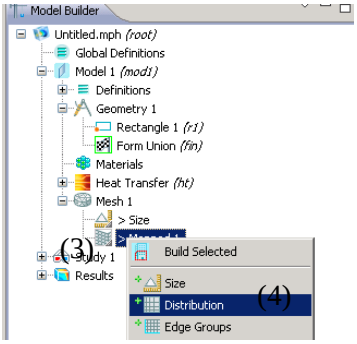
(4) Click the blue “Build All” icon to create the specified rectangle.

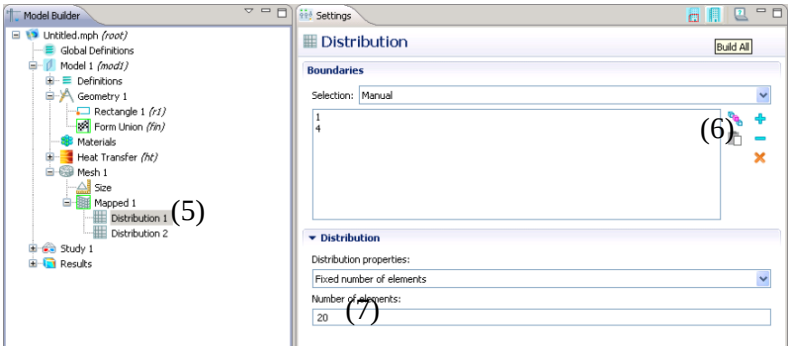


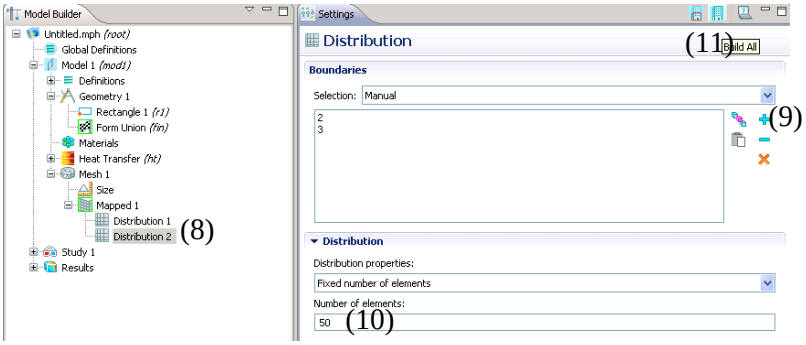

(5) Should result in the following geometry.

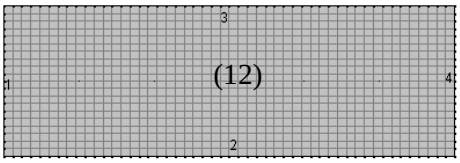
## Meshing



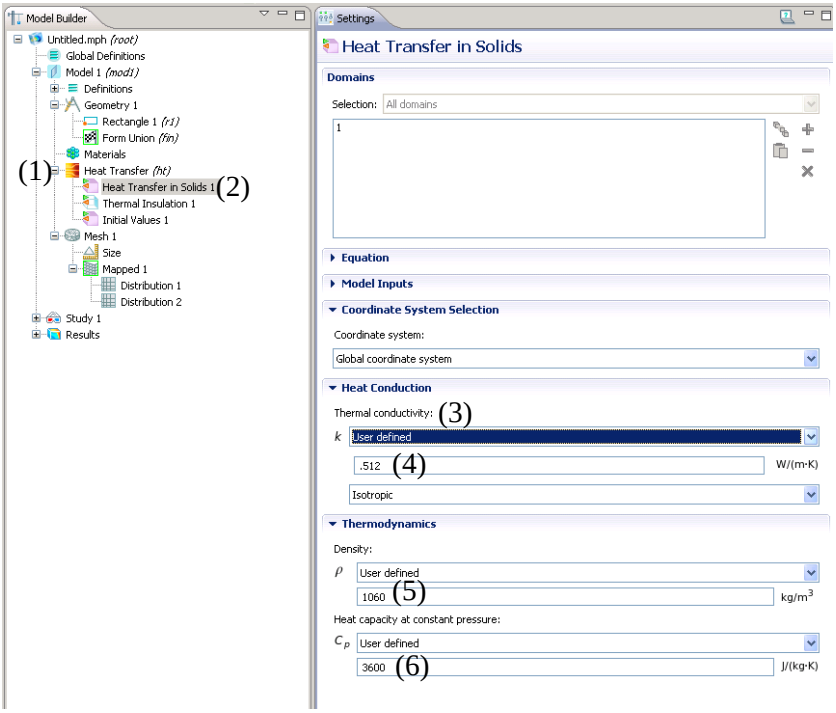



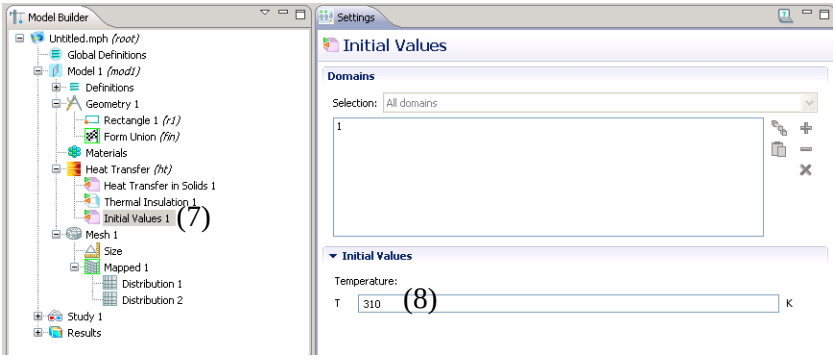



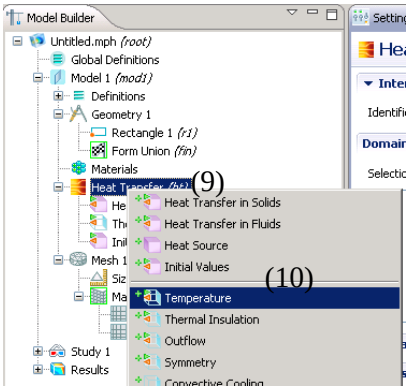




(1) Under **Model Builder**, right click on “Mesh 1”.
(2) Select “Mapped”.
(3) Under “Mesh1”, right click on “Mapped 1” and select “Distribution”.
(4) Repeat step 3 to end up with 2 “Distribution” options.
(5) Click on “Distribution 1”.
(6) Under **Graphics,** hold CTRL and click on boundary 1 and 4 (left and right boundaries) on the rectangle. Press + button to add.
(7) In **Distribution**, input 20 for “Number of elements”.
(8) Select “Distribution 2”.
(9) Hold CTRL and click on boundary 2 and 3 (top and bottom boundaries) on rectangle. Press + button to add to distribution.
(10) Input 50 for “number of elements”.
(11) Click blue “Build All” button to create a mapped mesh.
(12) The following is the mesh that should have been obtained.

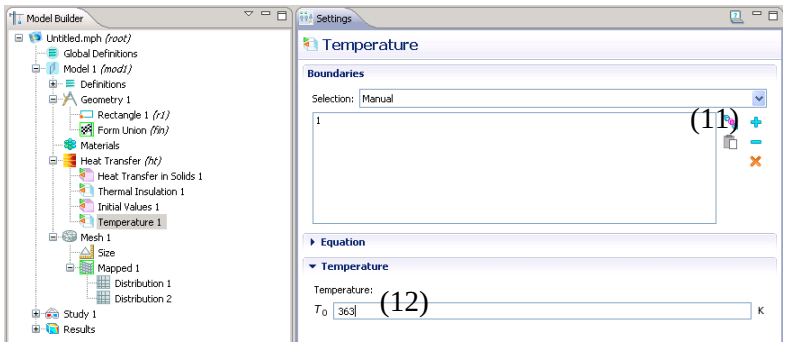
## Defining Material Properties and Parameters



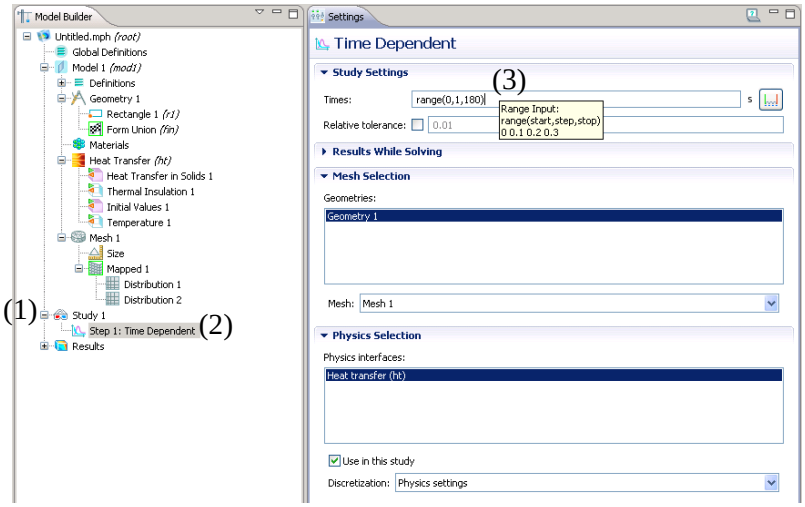

(1) In **Model Builder,** expand "Heat Transfer *(ht)*".
(2) Click on "Heat Transfer in Solids 1".
(3) Under **Heat Conduction,** select "User defined" for all properties (k, p, Cp).
(4) Input 0.512 W/(mK) for thermal conductivity.
(5) Input 1060 kg/m^3 for density.
(6) Input 3600 J/(kgK) for heat capacity.
(7) Select "Initial Values 1" under "Heat Transfer *(ht)*".
(8) Set initial temperature to 310K.
(9) Right Click "Heat Transfer *(ht)*".
(10) Select "Temperature" to add new constant temperature boundary condition.

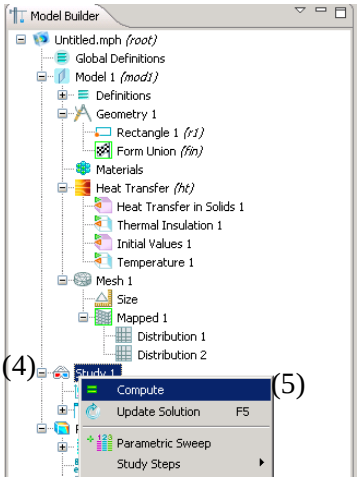



(11)Under **Graphics,** select boundary 1 and click + button to add boundary.

(12)Set boundary 1 temperature to 363K.

## Computation and Analysis



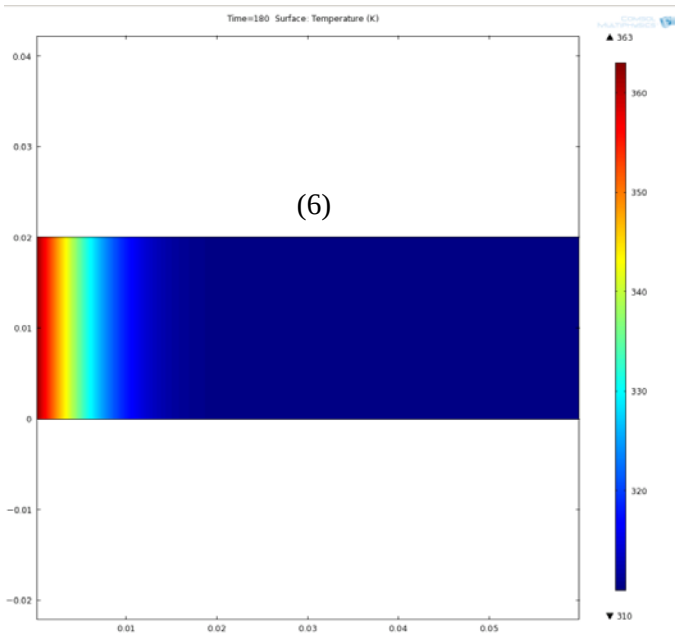

(1) Expand “Study 1”.

(2) Select “Step 1: Time Dependent”.

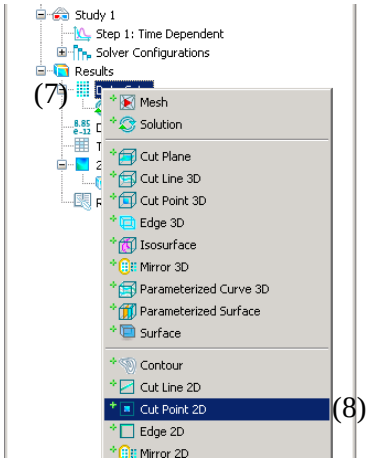
(3) In Study Settings, under Times input range (0,1,180).

(4) Right click on “Study 1”.

(5) Select “Compute”.

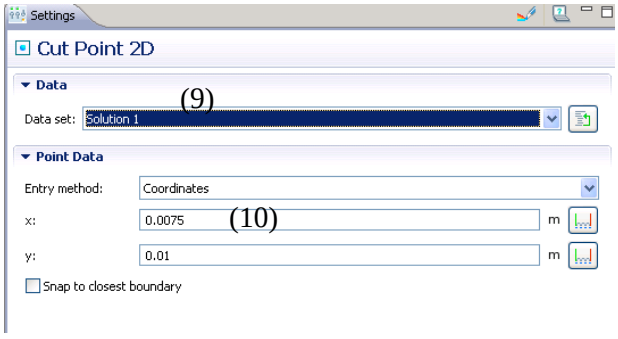



(6)




(7)

(8)




(9)

(10)

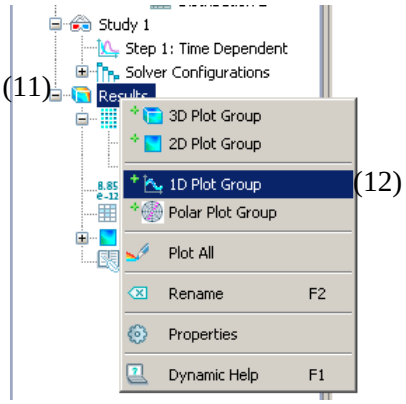
(6) Should result in a surface map at 180s.

(7) Right Click on “Data Sets” under Results

(8) Select “Cut Point 2D” located towards the middle of the options column.

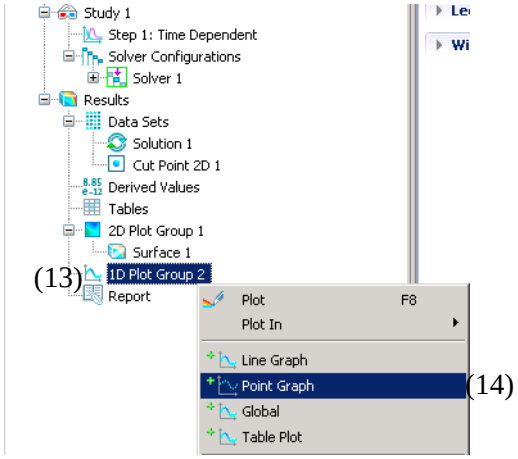
(9) In the “Settings” window, select “Solution 1” under pull down option for “Data set”.

(10) Input 0.0075m for the x-coordinate and 0.01m for y-coordinate.



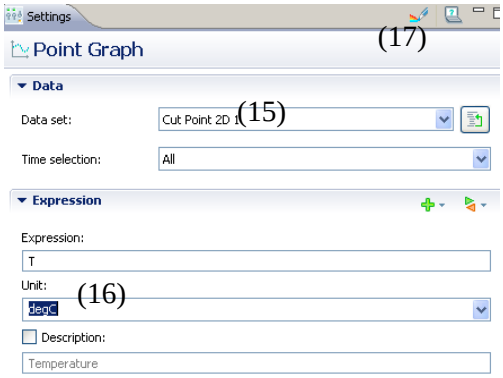

(11)

(12)




(13)

(14)

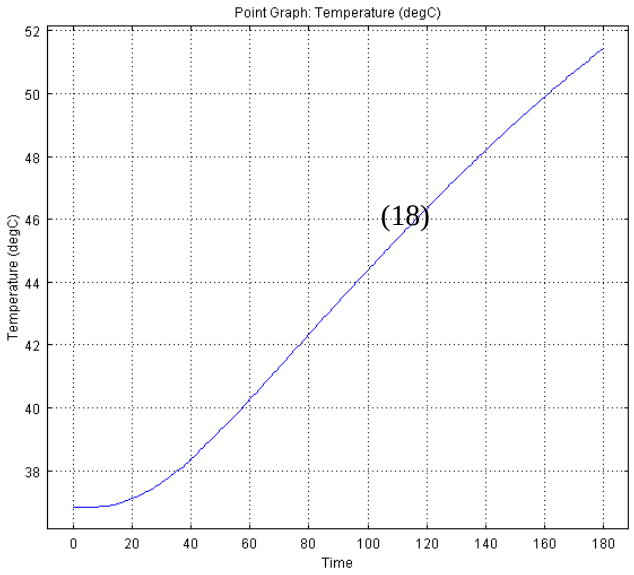



(17)

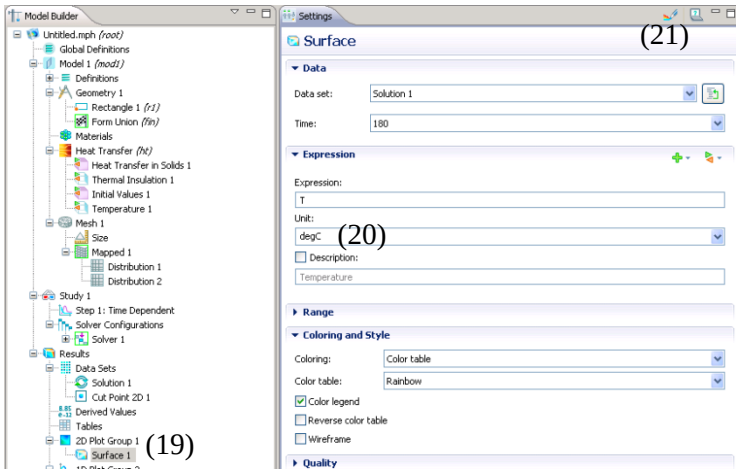
(15)

(16)

(11)Under **Model Builder** tab, right click on "Results".
(12)Select "1D Plot Group".
(13)Right click on "1D Plot Group 2".
(14)Select "Point Graph".
(15)In Settings, select "Cut Point 2D 1" from pull down options for Data Set.
(16)Under Expression choose "degC" from the drop-down menu.
(17)Click the Plot button.



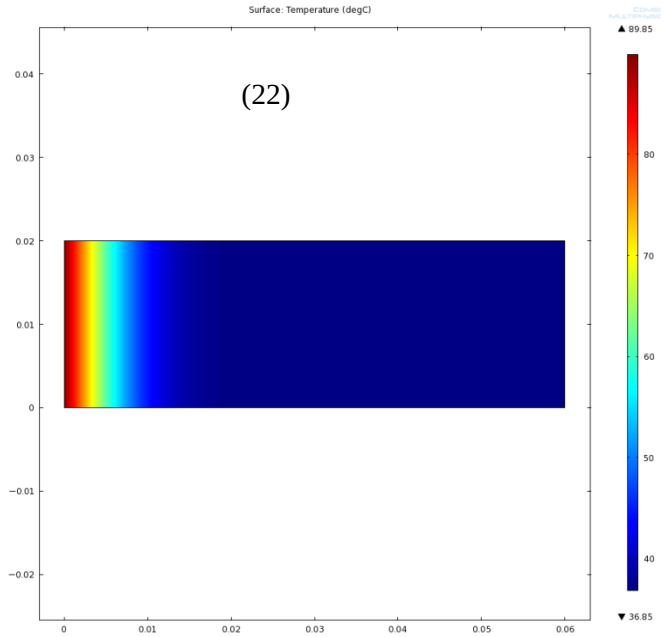

(18)




(19)

(20)

(21)




(22)

(18)Temperature profile at (0.0075,0.01).

(19)Expand “2D Plot Group 1” and Click on “Surface 1”.

(20)Under **Expression**, select “degC” from drop down menu for “Unit”.

(21)Click on the rainbow “Plot all” icon.

(22)Final surface plot in degree Celsius.

**Note: you can always return to the surface plot by clicking on “2D Plot Group 1”