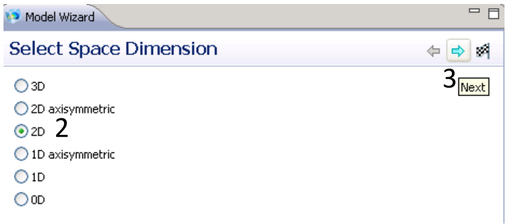
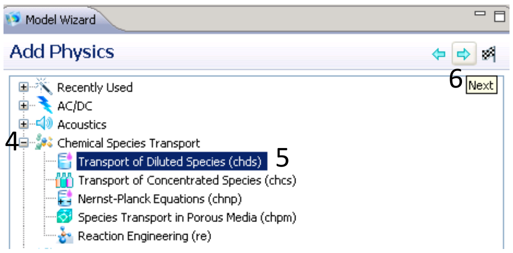
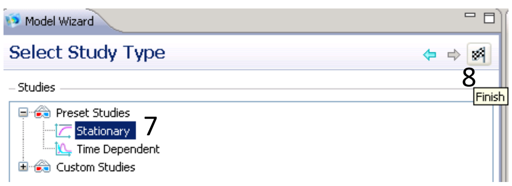
Problem Type Specification

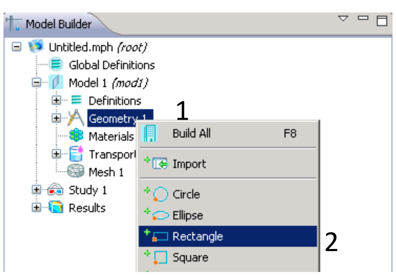


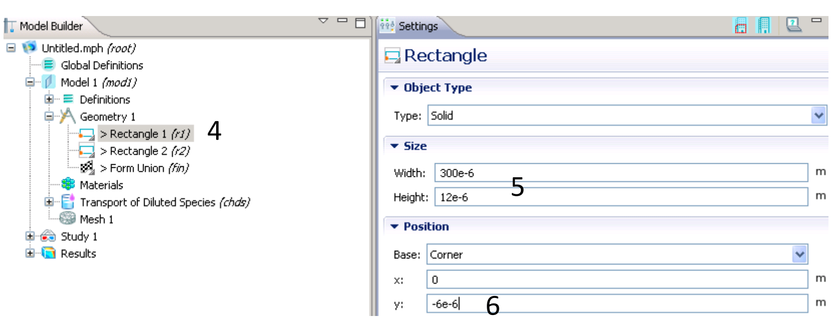




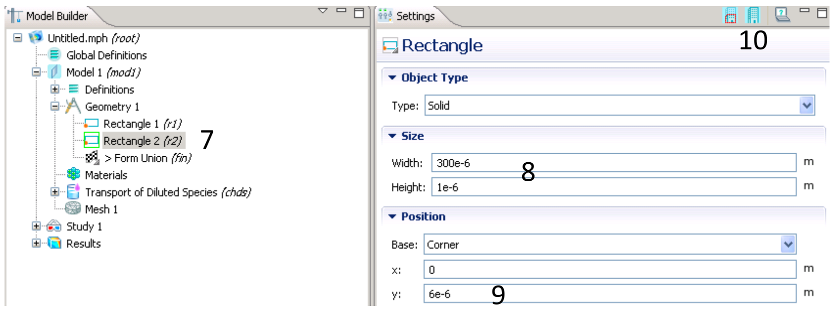
(1) Open COMSOL Multiphysics and safe file often to prevent lossing work.
(2) In **Select Space Dimension**, select 2D.
(3) Click on the blue next arrow.
(4) In **Add Physics**, expand "Chemical Species Transport".
(5) Double click on "Transport of Diluted Species (chds)".
(6) Click on the blue next arrow.
(7) Under **Select Study Type**, select "Stationary".
(8) Click on checkered flag to finish.

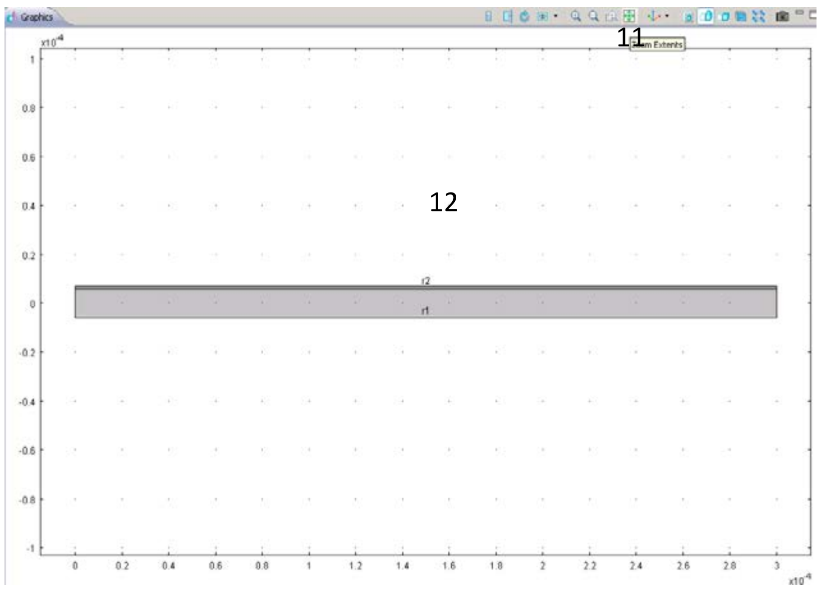
Geometry Creation





(1) Under **Model Builder**, right Click on "Geometry 1".
(2) Select "Rectangle".
(3) Repeat steps (1) and (2).
(4) Select "Rectangle 1 (r1)".
(5) Under **Settings**, input rectangle width as 300e-6 m and height as 12e-6 m.
(6) Input the y position as -6e-6 m and make sure base is set to "Corner".





(7) Select "Rectangle 2 (r2)".

(8) Under **Settings**, input rectangle width as 300e-6 m and height as 1e-6 m.

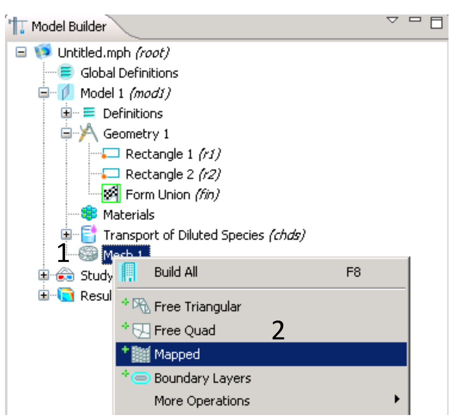
(9) Input the y position as 6e-6 m.

(10) Click the build "Build All" icon.

(11) Under **Graphics**, click on "Zoom Extents" icon to expand geometry.

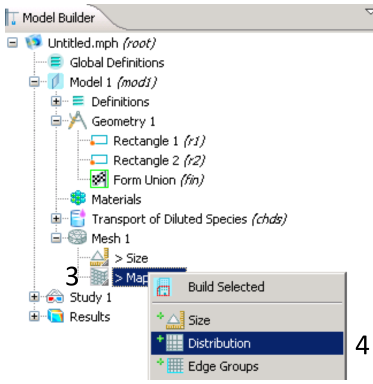
(12) The following geometry should result (contains 2 domains).

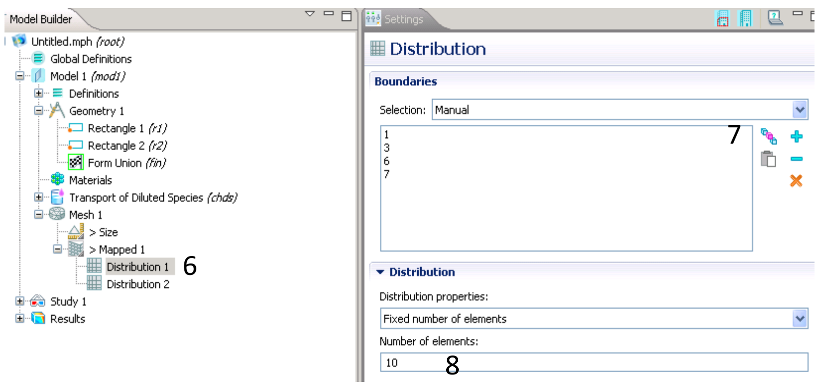
Meshing

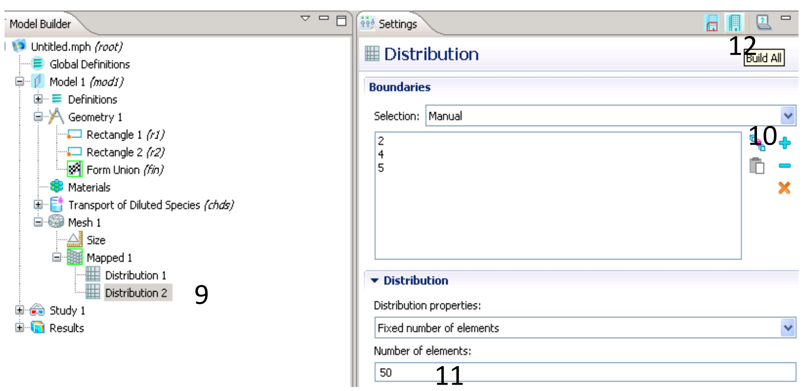


(1) Under **Model Builder**, right click on "Mesh1".

(2) Select "Mapped" mesh.







(3) Right Click on "Mapped 1".

(4) Select "Distribution".

(5) Repeat steps (3)&(4).

(6) Select "Distribution 1".

(7) Hold down CTRL and in the **Graphics** window click on boundaries 1, 3, 6, and 7. Click on the blue + icon to add boundaries to the distribution 1.

(8) Set "Number of elements" to 10 mesh units.

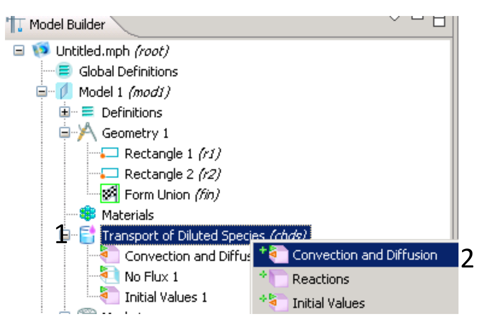
(9) Select "Distribution 2" under **Model Builder.**

(10)Hold down CTRL and click on boundaries 2,4, and 5 and click the blue + icon to add boundaries to distribution 2.

(11)Set "Number of elements" to 50 mesh units.

(12)Click Build All icon.

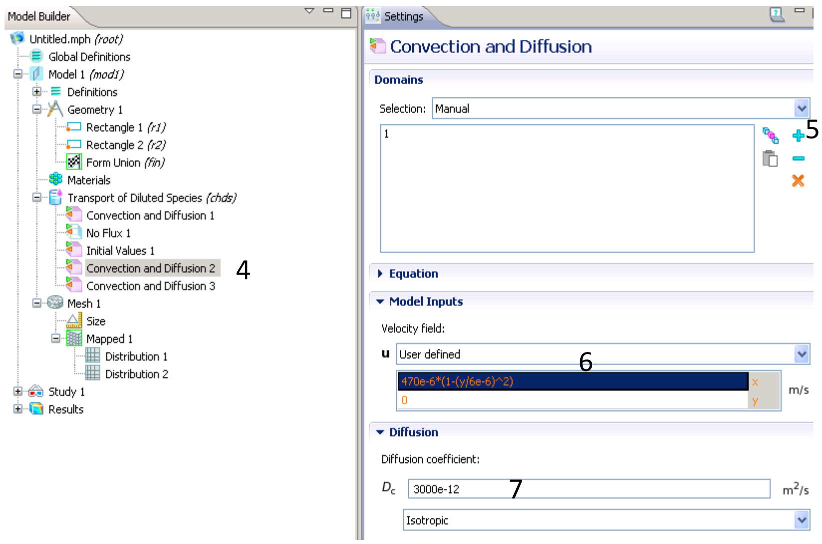
Subdomain and Boudary Settings

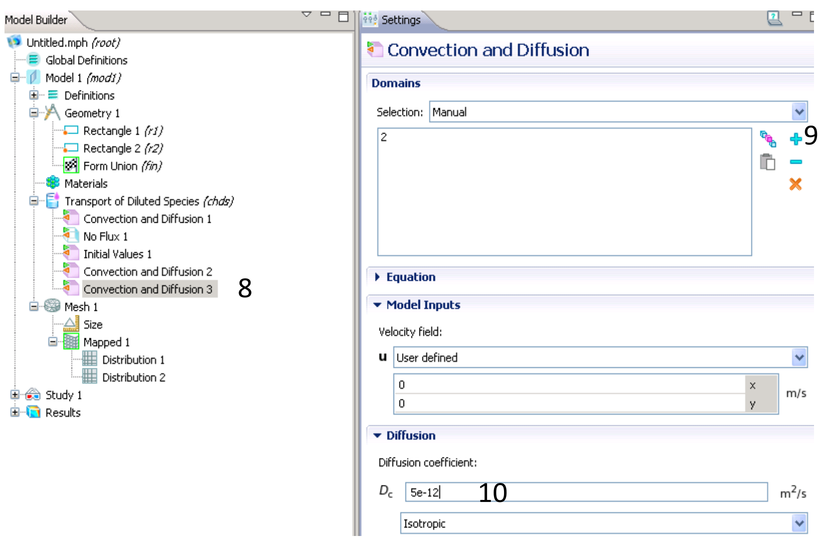


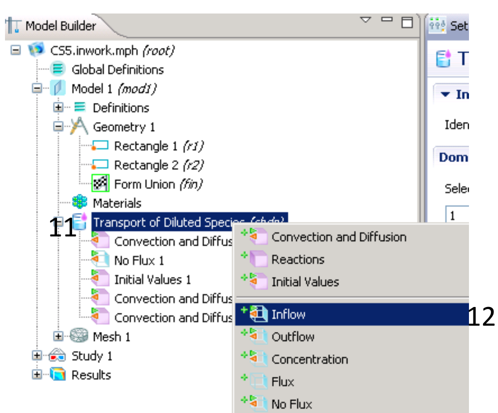
(1) Right click on "Transport of Diluted Species *(chds)*" under **Model Builder**.

(2) Select "Convection and Diffusion".

(3) Repeat steps (1) and (2).







(4) Click on "Convection and Diffusion 2".

(5) Click on subdomain 1 under **Graphics** window and click blue + icon.

(6) Under "Velocity field", input 470e-6*(1-(y/6e-6)^2) [m/s] for the x-direction.

(7) Input 3000e-12 [$m^2/2$] for Diffusion coefficient.

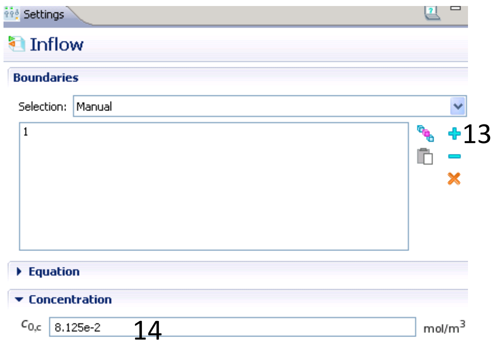
(8) Click on "Convection and Diffusion 3".

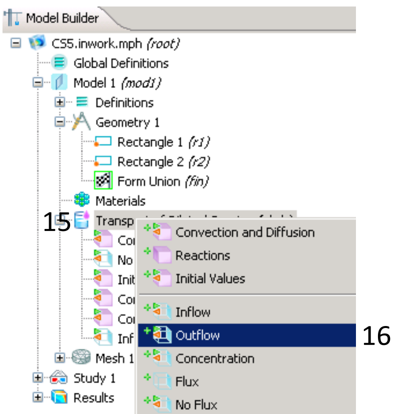
(9) Select subdomain 2 on **Graphics** window and click on the blue + icon.

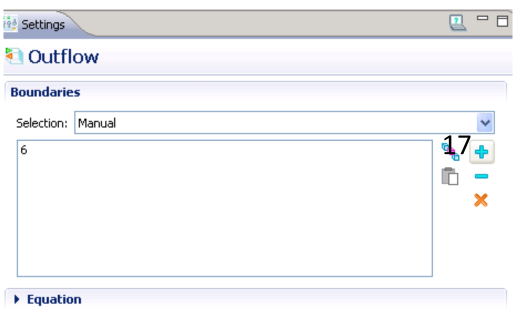
(10)Input 5e-12 [m^2/s] for "Diffusion coefficient".

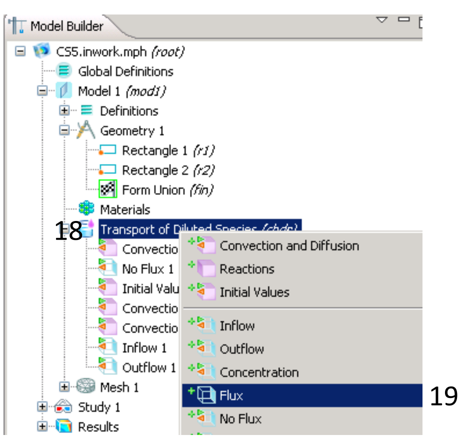
(11)Right click on "Transport of Diluted Species *(chds)*".

(12)Select "Inflow".









(13)Select boundary 1 from the **Graphics** window and click on blue + icon to add boundary condition.

(14)Set inflow concentration to 8.125e-2 [kg/m^3] (ignore COMSOL's units for concentration).

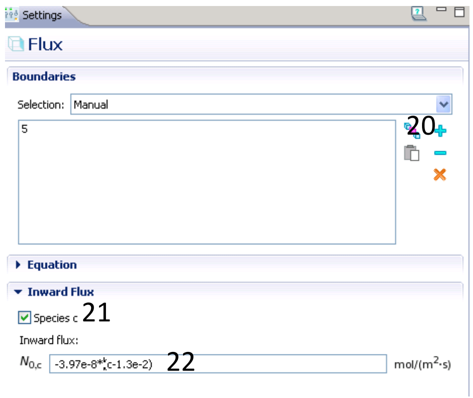
(15)Right click on "Transport of Diluted Species *(chds)*".

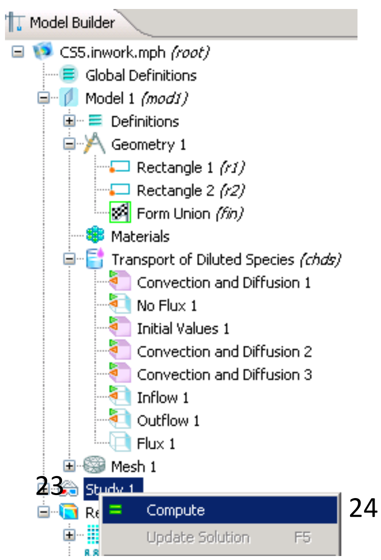
(16)Select on "Outflow".

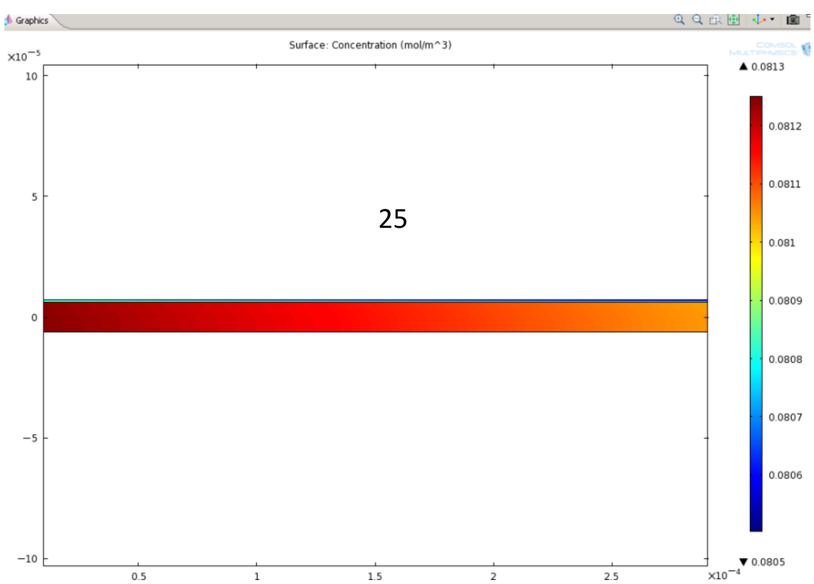
(17)Select boundary 6 and add as convective flux boundary condition.

(18)Right click on "Transport of Diluted Species *(chds)*".

(19)Select "Flux".







(20)Select on boundary 5 and add flux boundary condition.

(21)Check off box for "Species c".

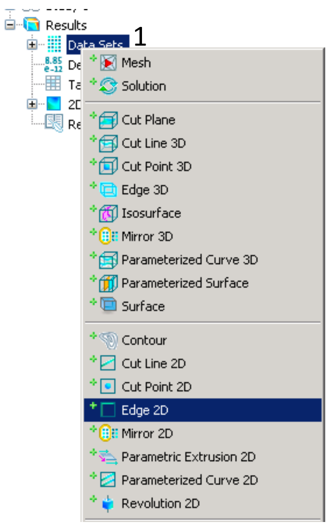
(22)Input -3.97e-8*(c-1.3e-2) [kg/(m^{2}s)] for inward flux.

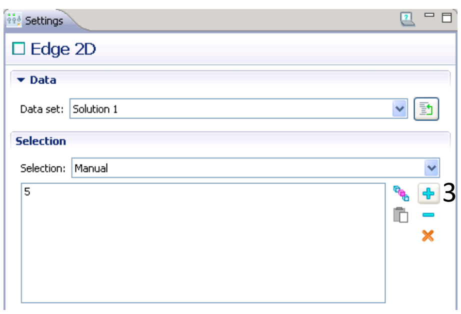
(23)Right Click on "Study 1".

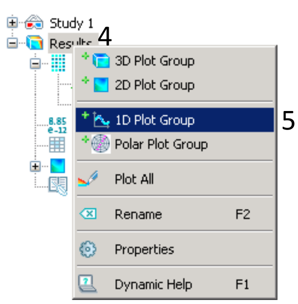
(24)Select "Compute".

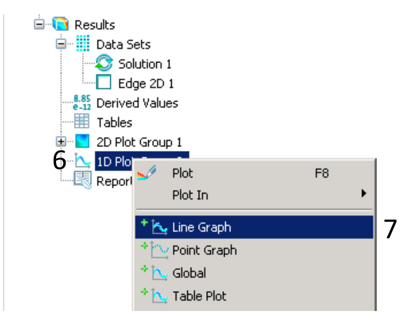
(25)The following surface plot should result.

Postprocessing

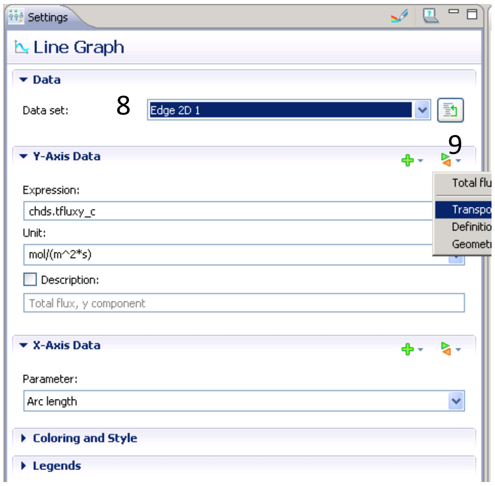


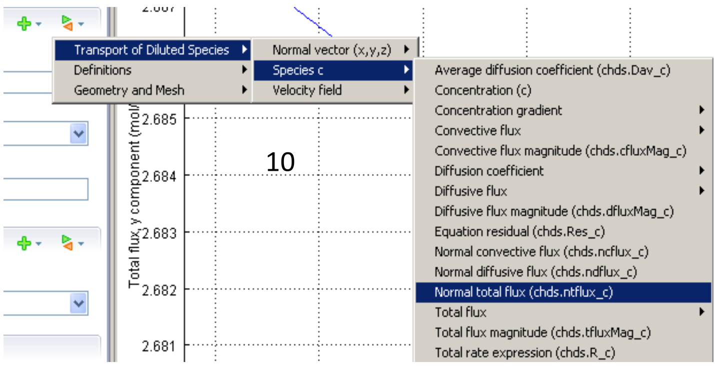


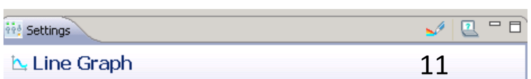


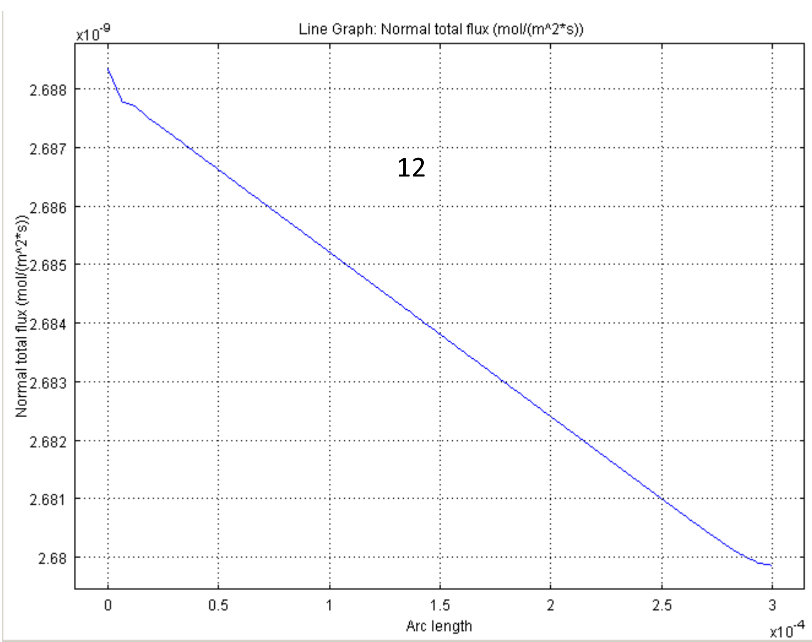


(1) Under "Results", right click on "Data Sets".
(2) Select "Edge 2D".
(3) Select boundary 5 from the **Graphics** window and click on blue add icon.
(4) Right Click on "Results".
(5) Select to add "1D Plot Group".
(6) Right click on "1D plot Group 2".
(7) Select to add "Line Graph".









(8) Under **Line Graph**, select "Edge 2D 1" from the pull down bar for "Data set".

(9) Change the expression to be evaluated by clicking on the green-orange up-down icon.

(10) "Transport of Diluted Species" >> "Species c">> "Normal Total flux (chds.ntflux_c)".

(11) Click the rainbow pencil to create plot.

(12) Following plot should result.