### Specifing the problem type



## **Creating the Geometry**



- Open COMSOL 4.1
   and save file to
   prevent loss of work.
- (2) Under Select Space Demension, select 2D axisymmetric.
- (3) Click on blue Next arrow.
- (4) Under Add Physics, expand "Chemical Species Transport".
- (5) Double click on"Transport of Diluted Species (chds)" to add new physics.
- (6) Click on blue Next arrow.
- (7) Under **Select Study Type**, select "Time Dependent".
- (8) Click checked flag to finish setting up.

- Under Model Builder, right click on "Geometry 1".
- (2) Select "Rectangle".



- (3) Under Settings, input
   0.03m for width(half
   of 6cm since we are
   working with
   axisymmetric) and
   0.0012m for height.
- (4) Click on blue "Build All" icon to build the desired geometry.
- (5) Right click on "Geometry 1".
- (6) Select "Point".
- (7) Input polar
  coordinate (0.024,
  0.0012) to mark the
  boundary of the
  patch.
- (8) Click on blue "Build All" icon.
- (9) The following geometry should appear in the Graphics window.

### Meshing



- Under Model
   Builder, right click on "Mesh 1".
- (2) Select "Mapped".
- (3) Right click on"Mapped 1".
- (4) Select "Distribution".
- (5) Repeat steps (3) and(4) 3 more times to end up with 4 Distributions.
- (6) Select "Distribution 1".
- (7) Hold CTRL and select boundary 1 and 5 on geometry from the Graphics window.
- (8) Click blue plus icon in the **Distribution** window to add boundaries to Distribution 1.
- (9) Set "Number of elements" to 5.

(7)



(10)Select "Distribution 2". (11)Select boundary 2 (bottom boundary), interface between skin and blood, and click on add icon. (12)Input 125 into "Number of elements". (13)Select "Distribution 3". (14)Select boundary 3 (top boundary), interface between patch and skin, and click on add icon. (15)Input 100 into "Number of elements". (16)Select "Distribution 4". (17)Select boundary 4 (skin not covered by patch) and click on add icon. (18)Input 25 into "Number of elements". (19)Click on blue "Build All" icon. (20)The following mesh should result with total of 625 mesh elements.

#### V 🗖 🗖 🙀 Settings Model Builder 2 - 🗆 🖃 🧐 Untitled.mph *(root)* ~ Selection: All domains Global Definitions 🖶 🚺 Model 1 (mod1) ₽ 1 °. Definitions A Geometry 1 Ŀ × 🗔 Rectangle 1 (r1) Point 1 (pt1) Form Union (fin) (1) Materials Transport of Diluted Species (chds) Convection and Diffusion 1 (2) Equation Model Inputs No Flux 1 initial Values 1 Velocity field: 🖻 🛞 Mesh 1 u User defined \* Size 0 r m/s 0 Distribution 1 7 Distribution 2 ▼ Diffusion 🔠 Distribution 3 Distribution 4 Diffusion coefficient: 🗄 🚓 Study 1 D<sub>c</sub> 1.11e-11 (3) 🗄 <u>वि</u> Results m²/s Isotropic ~ - - Citings T Model Builder 🖳 🖵 E 🖃 🧐 Untitled.mph *(root)* 🖲 Initial Values Global Definitions Model 1 (mod1) Domains E Definitions A Geometry 1 Selection: All domains - Rectangle 1 (r1) e. ÷ Point 1 (pt1) ľ -🛃 Form Union *(fin)* × 🏶 Materials 🖶 📑 Transport of Diluted Species *(chds)* l Convection and Diffusion 1 Axial Symmetry 1 ×. No Flux 1 Initial Values 1 (4) ▼ Initial Values 🖃 🎯 Mesh 1 Concentration: Size (5) c 0 mol/m<sup>3</sup> Distribution 1 - - D) 抗 Model Builder Settings Untitled.mph (root) Species 😑 Global Definitions 🖨 👖 Model 1 (modi) ▼ Interface Identifier 🗄 😑 Definitions Identifier: chds 🖶 🎢 Geometry 1 Domains Point 1 (pt1) Form Union (fin) Selection: All domains Materials (6) Transport of Dilute 🦄 Convection and Diffusion Convection and 🔸 Reactions Axial Symmetry Initial Values 된 No Flux 1 Tnitial Values 1 🖻 🛞 Mesh 1 \* 🚺 Outflow Size \* Concentration Distribution + Hux (7) No Flux Distribution 📰 Distributior 🔧 Symmetry Distributior 🔸 Flux Discontinuity 🗄 승 Study 1 \* Periodic Condition 🗄 🛅 Results \*🔄 Open Boundary Pairs

# **Defining Material Properties and Parameters**

- (1) Under Model
   Builder, expand
   "Transport of Diluted
   Species (chds)".
- (2) Select "Convection and Diffusion 1".
- (3) Under Settings window, set
  "Diffusion coefficient" as 1.11e-11 m^2/s.
- (4) Click on "Initial Values 1".
- (5) Make sure "Concentration" is 0.
- (6) Right click on"Transport of Diluted Species (chds).
- (7) Select "Flux" boundary condition.



- (8) Click on boundary 3 and click on blue plus icon to add boundary.
- (9) Under "Inward Flux", check box for "Species c".
- (10)Set "Inward flux" to 8.849e-7 mol/(m<sup>2</sup>s).
  (11)Right click on
  - "Transport of Diluted Species (chds).
- (12)Select
  - "Concentration" boundary condition.
- (13)Click on boundary 2
- and click on blue plus icon to add
  - boundary.
- (14)Under "Concentration",
  - check box for
  - "Species c".
- (15)Set boundary concentration to equal 0 since all drug that diffuses this far will be cleared by blood flow.

### **Computation and Post-Modifications**



- (1) Expand "Study 1".
- (2) Select "Step 1: Time Dependent".
- (3) Under TimeDependent, inputrange(0,3600,604800) for "Times".
- (4) Right click on "Study 1".
- (5) Select "Compute".
- (6) The following surface plot should result for solution at 604800s.



- (7) Under "Results", right click on "Data Sets".
- (8) Select "Cut Point 2D".
- (9) Under Settings, input 14.4e-3 for r-value and 0.6e-3 for zvalue.
- (10)Right click on "Results".
- (11)Select "1D Plot Group".
- (12)Right click on "1D
  - Plot Group 2".
- (13)Select "Point Graph".



(14)Under Point Graph choose "Cut Point 2D 1" from the drop down menu for "Data set". (15)Click the plot icon (rainbow pencil icon). (16)The following concentration profile should result for polar coordinate (14.4e-3, 0.06e-3). (17) Under "Results" select "2D Plot Group 1". (18)Change "Time" to 2.592e5 using the drop down menu. (19)Click the Plot icon. (20)The following surface plot should result of the solution at 2.592e5 seconds.