

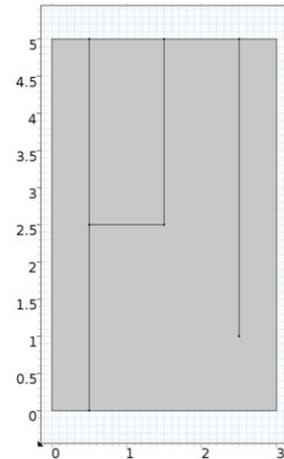
XXXXX - Tutorial XX Toolbox

1 Set up a Multiphysics model with fractures implemented as weak contributions.

1.1 Setup of the flow model

1. Start Multiphysics.
2. In the model wizard, select 2D.
3. Select Mathematics → PDE Interfaces → Coefficient Form PDE (*c*). Press the 'Add' button and change the Dependent Variable name to be 'H'. Choose Head (m) as Dependent variable and 1/s as Unit for the Source term.
4. Press the right arrow key and select a Stationary study. Press 'Done'.
5. In the model builder, right click on Geometry 1 and select Rectangle. Enter a Width of 3m and a Height of 5m. Right click on Geometry 1 and select Build all.
6. Add several line segments. Right click on Geometry 1 and Bézier Polygon and choose 'Open curve' as type. With 'Add Linear', you can enter the coordinates of each line (you will need to create several polygons). Or you can use the Draw Line option in the Geometry menu (also with an Open curve) to build the lines.

X1	Y1	X2	Y2
0.5	0	0.5	5
0.5	2.5	1.5	2.5
1.5	2.5	1.5	5
2.5	1	2.5	5



7. When you have finished that, right click on Geometry and choose Build all. The geometry should look as shown on the right then.
8. Right click on Component1 → Definitions and create a Variables object. Click on Variables 1 and press the Load from File button. Load the provided 'Variables_Weak' file. Have a look at the variables.
9. Under Coefficient Form PDE (*c*) → Coefficient Form PDE 1, choose the Diffusion Coefficient *c* to be isotropic and equal to K_m . All other coefficients should be 0.
10. Rename the Coefficient Form PDE (*c*) to Flow.

1.2 Setup of the transport model

1. Add another PDE: Right click on Component1 and Add Physics and select Coefficient Form PDE (*c*). Press Add to Selection. Choose Concentration (kg/m^3) as Dependent variable and change the name to C_m . The Unit of the Source term quantity is $\text{kg}/(\text{m}^3 \cdot \text{s})$.

- Go back to the Model Builder and enter the equation. The Diffusion coefficient c is isotropic and equal to $Dm \cdot n$. The mass coefficient d_a is n and the Conservative Flux Convection Coefficient α is equal to $-v_x \cdot n$, $-v_y \cdot n$. All other coefficients should be 0.
- Rename this equation to Transport.

1.3 Initial and boundary conditions

- Enter fixed head values of 1m at the top and 0m at the bottom of the domain. The other flow boundaries should remain as zero-flux boundaries.
- Set up the mesh. Right click on Mesh 1 → Free Triangular. Right click on Mesh 1 and press 'Build All'. If you later on observe oscillations, try to refine the mesh (especially at fractures and at places with strong gradients).
- Click on Study 1 → Step1: Stationary and uncheck Solve for Transport (c2). Right click on Study 1 and press compute. View the surface plot of H without fractures.
- For transport, set an outflow condition at the lower boundary. For that purpose, add a Flux/Source condition, add the lower boundary segments and set $g = -(v_x \cdot n_x + v_y \cdot n_y) \cdot C_m \cdot n$. All other boundaries obtain no flow conditions.

2 Set up of weak contributions

2.1 Weak contribution of the flow

Add a new set of Variables to the Definitions and specify the weak equations. The following expression is the weak formulation for the flow in the fracture, with the tangential derivatives of the head along the fracture, HTx and HTy:

$$\text{weak_frac_h} = \text{test}(\text{HTx}) \cdot (-Kf \cdot \text{HTx} \cdot \text{Ap}) + \text{test}(\text{HTy}) \cdot (-Kf \cdot \text{HTy} \cdot \text{Ap})$$

2.2 Weak contribution of the contaminant transport

- With Cmt being the time derivative of the concentration, the time-dependent term in the transport equation is expressed as $d\text{weak_frac_c} = \text{test}(Cm) \cdot \text{Ap} \cdot Cmt$
- The fluxes and the source and sink term is defined as $\text{weak_frac_c} = \text{test}(CmTx) \cdot (-Df \cdot CmTx \cdot \text{Ap} + vfx \cdot Cm \cdot \text{Ap}) + \text{test}(CmTy) \cdot (-Df \cdot CmTy \cdot \text{Ap} + vfy \cdot Cm \cdot \text{Ap}) + \text{test}(Cm) \cdot (-k \cdot Cm \cdot \text{Ap})$
- To employ the weak contributions in the model, right click on the Flow (c) and add a More → Weak Contribution for boundaries (second block). Enter weak_frac_h there. Select the interior lines you created in 1.1.7.
- Right click on Study 1 and press Compute. Observe the changes in the heads.
- Add also a Weak Contribution to Transport (c). Enter $d\text{weak_frac_c} + \text{weak_frac_c}$ there and select the fractures.

6. The transport in the fracture requires an outflow condition at the left fracture, which ends on the lower boundary. Right click on Transport (c_2) and add a Points \rightarrow Weak Contribution. Select the point where the fracture meets the lower boundary and enter $\text{test}(C_m)*(-K_f*(H_x+H_y)*A_p*C_m)$ as Weak expression.
7. A high concentration at the inlet of the fractures is set as constraint via a right click on Transport (C_2), Points \rightarrow Pointwise constraint. Then, select the three points where the fractures enter at the upper boundary. Enter $C_m - 10$ there. This will fix the concentration at these points to a value of 10 kg/m^3 .

2.3 Add a study and solve the transport problem

1. Right click on the (root) node and press Add Study. Uncheck Solve for Coefficient Form PDE (c) and choose a Time Dependent study type. Press the Add Study button and close the Add Study Dialog.
2. Go to Study 2 \rightarrow Step1: Time Dependent and uncheck Flow (c)
3. Select Values of Dependent Variables \rightarrow Variables not solved for, change Method to be 'Solution' and Solution to be 'Study 1, Stationary'.
4. Enter $\text{range}(0,200,5000)$ in the Times box at the top and select d (days) as Time unit. This tells the solver that the maximum simulation time is 5000 days and that the solution should be output every 200 days.
5. Right click on Study 2 and select 'Compute'.

2.4 View results

1. Examine the plot of concentration.
2. Under Export, add a player and select it, choose 2D Plot Group 3 as Subject. Do a right-click on the Player 1 and choose Generate Frame. Examine how the contaminant plume moves through the model.