Flow123d tutorial 1 – “1D column”

Contents

The first example is inspirited a real locality of a water treatment plant tunnel Bedřichov in the granite rock massif. There is the particular seepage site 23 m under the surface and it has very fast reaction on rainfall events. Real data of discharge and concentration of stable isotopes are used.

1 Geometry and boundary conditions

It is considered a pseudo one-dimensional model in the range 10×23 m of the atmospheric pressure on the surface and on the bottom and no flow boundary condition on the edges (Figure 1a).

2 Flux field computation

2.1 Set of model mesh

In control file is important a sensitive of spaces from beginning of row. It is mandatory to hang the structure of spaces in control file (“flow_gmsh_flux.yaml”). The spaces divide section of problem setting. Mesh file comes from “./input” folder.

```
mesh:
  mesh_file: ./input/mesh_ex1.msh
```

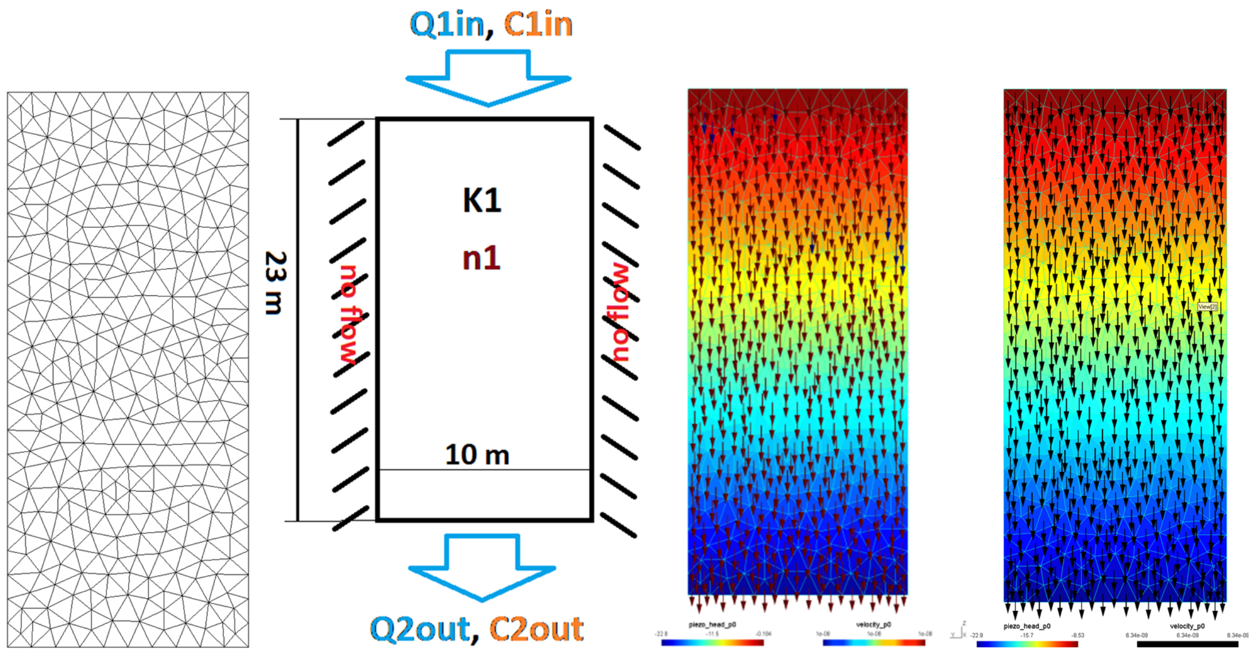

Figure 1: a) the geometry; b) the boundary condition of the example 1. c) Results of piezometric head and flux in gmsh software from file ex_1-flux.msh.

In this example we solve problem in Darcy flow and it is set at row-file 12 in “flow_gmsh_flux.yaml”.

```
flow_equation: !Flow_Darcy_MH
```

NOTE: The software Flow123d creates new folder for all results which consist from controlling file and appendix “.out”.

2.2 Set of model parameters

For the rock massif (“- region: rock”) we prescribed the hydraulic conductivity K = 1e-8 m/s. This value is typical for the granite rock massif. The cross-section parameter is set on the 1 m width.

Code illustration: prescription of hydraulic parameters input_fields:

```
input_fields:
  - region: rock
    conductivity: 1e-8
    cross_section: 1
  - region: dirichlet_boundary
    bc_type: dirichlet
    bc_pressure: 0
Control file: "flow_gmsh_flux.yaml"
```

2.3 Results

The results of computation are in the file “water_balance.txt”. The input flux on the surface is 1 $\times$ 10-7 and the output flux on the tunnel is -1 $\times$ 10-7 (Table 1).

“time”	“region”	“quantity [m(3)]”	“flux”	“flux_in”	“flux_out”
0	“rock”	“water_volume”	0	0	0
0	“.surface”	“water_volume”	1e-07	1e-07	0
0	“.tunnel”	“water_volume”	-1e-07	0	-1e-07
0	“IMPLICIT BOUNDARY”	“water_volume”	2.58e-26	6.46e-26	-3.87e-26

Table 1: Results in “water_balanced.txt” (edited table, no whole file).

2.4 Variants

In the main YAML file we can change some parameters for an investigation of the model behaviour.

2.4.1 Infiltration

First we change the atmospheric pressure on the surface to the more realistic infiltration 200 mm/yr (= 6.34e-9 m/s).

Code illustration: prescription of the flux on the surface

```
  - region: dirichlet_boundary
    bc_type: dirichlet
    bc_pressure: 0
Control file: "flow_gmsh_infiltration.yaml"
```

The results are in the file “water_balanced.txt” again. We can see that the value of the input and output flux changed to 6.34e-8. The visual results are similar to the

3 Transport model

The numerical diffusion is used for this example. The ordering equation is set at row 38 and 39:

```
solute_equation: !Coupling_OperatorSplitting
  transport: !Solute_Advection_FV
```

The boundary condition of concentration is prescribed on the surface region (rows 40-43):

```
input_fields:
  - region: .surface
    bc_conc: !FieldFormula
      value: 100
```

The name of substance was “A” for this example.

```
substances:
  - A
```

The output time step of printout was set in section output_stream:

```
output_stream:
  time_step: 1e7
```

And the end time was set in section time:

```
time:
  end_time: 1e10
```

3.1 Transport model - results

The balanced results of the transport computation are in the output folder in the file "mass_balance.txt". The concentration is depicted on the Figure 2. Through the ".surface", the concentration is still identical (6 × 10-6). Through the based marked ".tunnel", the concentration is zero at the beginning and then is changed around 100 years to the opposite value of inflow -6 × 10-6. The selected part of numerical results of mass is in the Table 2. The figure 3 depicts results from file "mass_balance.txt" for mass transported through the boundaries ".surface" and ".tunnel" and in the volume of model "rock".

Note: Each model boundary is mandatory to assign with dot in a mesh file: ".surface".

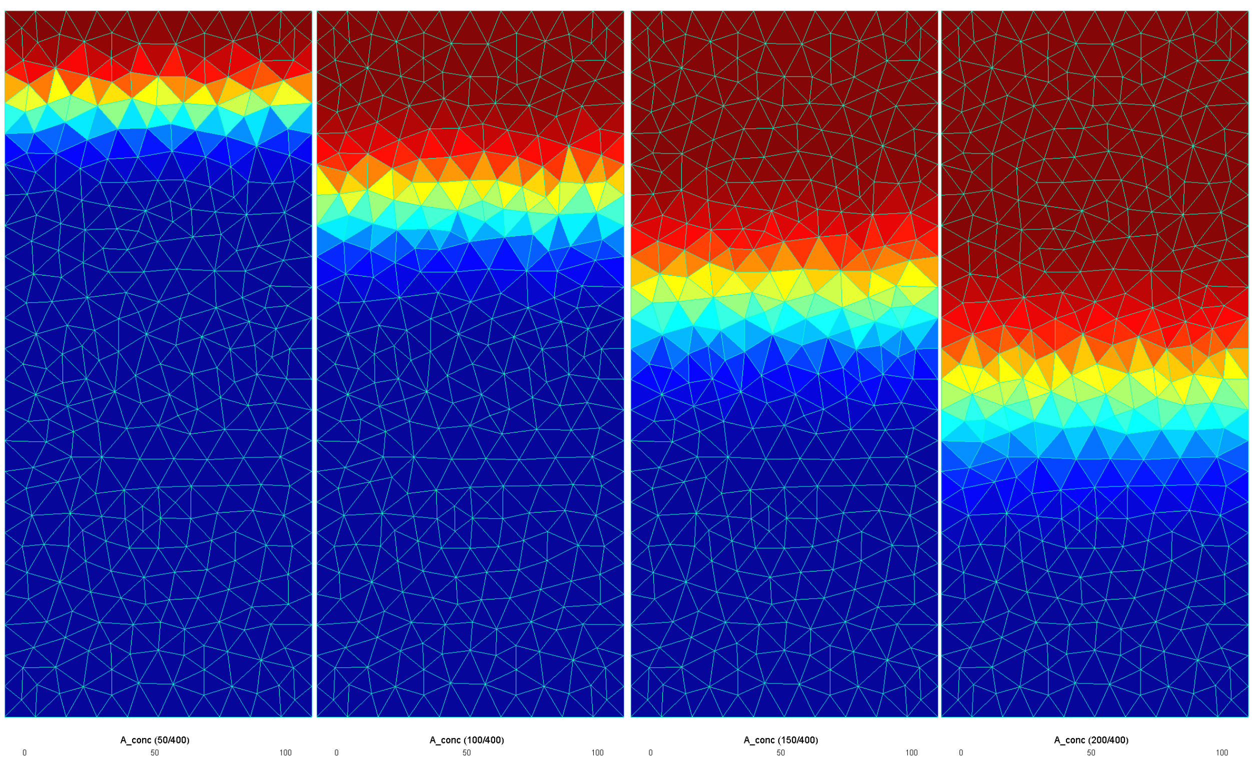

Figure 2: Results of transport from "mass_balance.txt".

time	region	quantity [kg]	flux	flux_in	flux_out	mass	error
3.96E+09	rock	A	0	0	0	22544.1	0
3.96E+09	.surface	A	6.34E-06	6.34E-06	0	0	0
3.96E+09	.tunnel	A	-4.88E-06	0	-4.88E-06	0	0

time	region	quantity [kg]	flux	flux_in	flux_out	mass	error
3.96E+09	IMPLICIT BOUNDARY	A	-1.02E-19	0	-1.02E-19	0	0
3.96E+09	ALL	A	1.46E-06	6.34E-06	-4.88E-06	22544.1	-7.39E-10
3.97E+09	rock	A	0	0	0	22558.7	0
3.97E+09	.surface	A	6.34E-06	6.34E-06	0	0	0
3.97E+09	.tunnel	A	-4.92E-06	0	-4.92E-06	0	0
3.97E+09	IMPLICIT BOUNDARY	A	-1.02E-19	0	-1.02E-19	0	0
3.97E+09	ALL	A	1.42E-06	6.34E-06	-4.92E-06	22558.7	-7.53E-10

Table 2: Illustration of the results in "water_balanced.txt" – selected column in two time steps (edited table, no whole file).

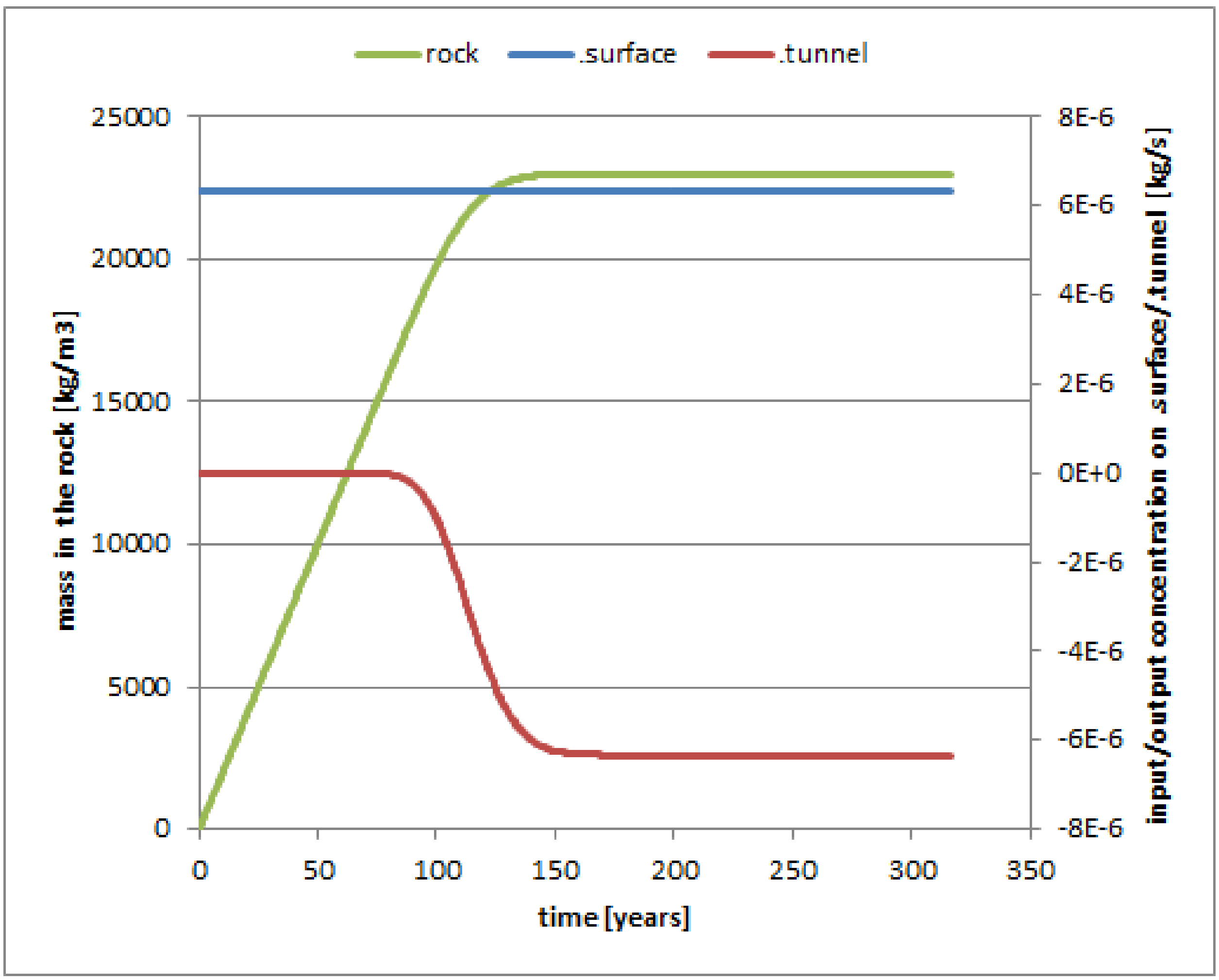

Figure 3: Results of transport in four time moments (1 time step = 3.8 months)

4 Conclusion

On the naive two-dimensional model the hydraulic and the transport model computation was shown.


```
flow123d_version: 1.8.9
problem: !Coupling_Sequential
  description: Example 1 of real locality - column 1D model
  mesh:
    mesh_file: ./input/mesh_ex1.msh
    regions:
      - !Union
        name: dirichlet_boundary
        regions:
          - .surface
          - .tunnel
  flow_equation: !Flow_Darcy_MH
    nonlinear_solver:
      linear_solver: !Petsc
        a_tol: 1.0e-15
        r_tol: 1.0e-15
    input_fields:
      - region: rock
        conductivity: 1e-8
        cross_section: 1
      - region: dirichlet_boundary
        bc_type: dirichlet
        bc_pressure: 0
    balance: true
    output:
      output_stream:
        file: ./ex_1-flux.msh
        format: !gmsh
          variant: ascii
      output_fields:
        - piezo_head_p0
        - pressure_p0
        - pressure_p1
        - velocity_p0
```