

# Flow123d tutorial 1 – “1D column”

## Contents

<b>1</b>	<b>Description</b>	<b>1</b>
<b>2</b>	<b>Input</b>	<b>1</b>
2.1	Setting the computational mesh . . . . .	2
2.2	Setting the model and physical parameters . . . . .	2
2.3	Setting solver parameters . . . . .	3
2.4	Setting output . . . . .	3
<b>3</b>	<b>Results</b>	<b>4</b>
<b>4</b>	<b>Variant</b>	<b>4</b>
<b>5</b>	<b>The control file</b>	<b>4</b>

## 1 Description

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

The user will learn how to:

- Set up the mesh and flow model input parameters;
- Set up the solver and output parameters.

A pseudo one-dimensional model is considered in the range  $10 \times 23$  m with the atmospheric pressure on the surface and on the bottom, and no flow boundary condition on the edges (Figure 1).

## 2 Input

The model settings are given in the control file, which is in YAML format. Every line contains one parameter and its value(s). The indentation of lines is important, since it indicates the section to which the parameter belongs.

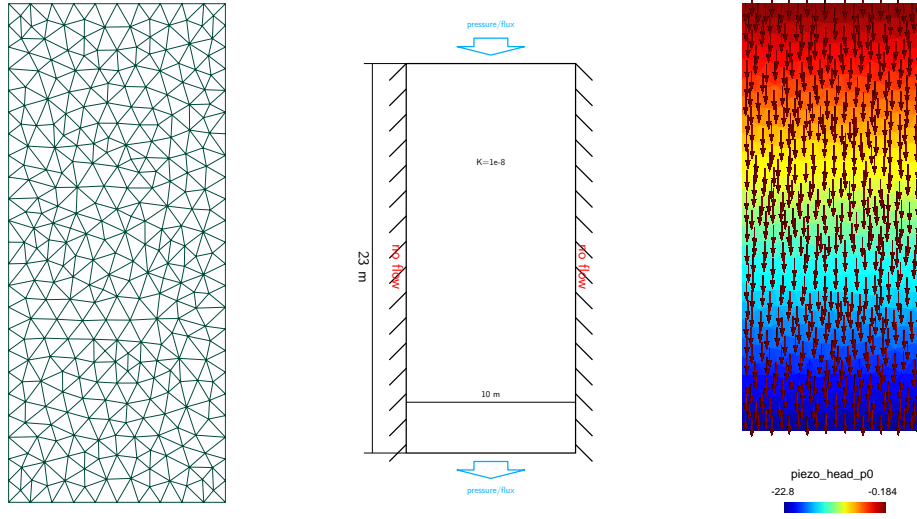


Figure 1: a) the mesh; b) the boundary conditions; c) computed piezometric head and flux.

## 2.1 Setting the computational mesh

The mesh file can be generated using the software [GMSH](#). It has to contain:

- point coordinates;
- simplicial elements (lines, triangles, tetrahedra). Elements of lower dimensions represent fractures or channels;
- physical regions (groups of elements, labeled either by numerical id or string caption). Names of regions defining boundary have to start by a dot;

The mesh file is specified by the following lines:

```
mesh:
  mesh_file: ./01_mesh.msh
```

Here the leading “./” denotes the directory where the control file is placed.

## 2.2 Setting the model and physical parameters

In this example we use the Darcy flow model, which is set by:

```
flow_equation: !Flow_Darcy_MH
```

Note: The equation name consists of three parts: physical process (flow), mathematical model (Darcy) and numerical method (MH = mixed hybrid finite element method).

The bulk parameters and boundary conditions are defined in the section `input_fields`. For the rock massif (- `region: rock`) we prescribe the hydraulic conductivity  $K = 10^{-8}$  m/s (typical value for the granite rock massif):

```
input_fields:
  - region: rock
    conductivity: 1e-8
```

We prescribe the atmospheric pressure both at the surface and the tunnel:

```
- region: .tunnel
  bc_type: dirichlet
  bc_pressure: 0
- region: .surface
  bc_type: dirichlet
  bc_pressure: 0
```

If no boundary condition is given then the default “no flow” is applied.

## 2.3 Setting solver parameters

For the solution of the linear algebra problem we have to specify solver type and tolerances for controlling the residual. In `flow_equation` we can use either `Petsc` solver which performs well for small and moderate size problems, or `Bddc` (a scalable domain decomposition solver). Two stopping criteria can be given: absolute and relative tolerance of residual.

```
nonlinear_solver:
  linear_solver: !Petsc
  a_tol: 1e-15
  r_tol: 1e-15
```

The key `nonlinear_solver` has further parameters which play role in other (nonlinear) flow models.

## 2.4 Setting output

In the section `output` we define the file name and type (supported types are `gmsh` and `vtk`, which can be viewed by GMSH, ParaView, respectively) to which the solution is saved:

```
output:
  output_stream:
    file: flow.msh
    format: !gmsh
    variant: ascii
```

The list of fields (solution components, input fields etc.) to be saved is specified by:

```
output_fields:
  - piezo_head_p0
  - pressure_p0
  - pressure_p1
  - velocity_p0
```

The above code can be alternatively written in a more compact form, namely

```
output_fields: [piezo_head_p0, pressure_p0, pressure_p1, velocity_p0]
```

In addition to the output of solution, Flow123d provides computation of balance of fluid volume, flux through boundaries and volume sources. This is turned on by

```
balance: true
```

### 3 Results

The results of computation are generated to the files `flow.msh` and `water_balance.txt`. From the balance file, one can see that 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, extract from the file).

### 4 Variant

Control file `02_column_transport.yaml` contains modified boundary conditions and solute transport model for the same physical problem.

### 5 The control file

Below is the complete YAML source.

```
flow123d_version: 1.8.9
problem: !Coupling_Sequential
  description: Example 1 of real locality - column 1D model
  mesh:
    mesh_file: ./01_mesh.msh
  flow_equation: !Flow_Darcy_MH
  input_fields:
    - region: rock
      conductivity: 1e-8
    - region: .tunnel
      bc_type: dirichlet
      bc_pressure: 0
    - region: .surface
      bc_type: dirichlet
```

```
    bc_pressure: 0
nonlinear_solver:
  linear_solver: !Petsc
  a_tol: 1e-15
  r_tol: 1e-15
output:
  output_stream:
    file: flow.msh
    format: !gmsh
    variant: ascii
  output_fields:
    - piezo_head_p0
    - pressure_p0
    - pressure_p1
    - velocity_p0
balance: true
```