

Technical university of Liberec  
Faculty of mechatronics, informatics  
and interdisciplinary studies

**FLOW123D**

version 1.6.0

Documentation of file formats  
and brief user manual.

**Liberec, 2010**

**Acknowledgement.** This work was realized under the state subsidy of the Czech Republic within the research and development project “Advanced Remediation Technologies and Processes Center“ 1M0554 – Program of Research Centers PP2-D01 supported by Ministry of Education.

# Flow123D

## Introduction

Flow123D is a software for simulation of water flow, solute transport and sorption in a heterogeneous porous and fractured medium. In particular it is suited for simulation of underground processes in a granite rock masive. The program is able to describe explicitly processes in 3D medium, 2D fractures, and 1D chanel and exchange between those dimensions. The computational mesh is therefore collection of 3D tetrahedrons, 2D trinagles and 1D line segments.

The water flow model assumes a saturated medium described by Darcy law. For discretization we use mixed hybrid FEM. This version allows only calculation of steady water flow.

The solute transport model can deal with several dissolved substances. It contains non-equilibrium dual porosity model, i.e. exchange between mobile and immobile pores. There is also model for several types of sorption in both the mobile and immobile zone. The imlemented sorption models are linear sorption, Freundlich isotherm and Langmuir isotherm. The solute transport model uses finite volume discretization with upwinding in space and explicit Euler discretization in time. The dual porosity and the sorption are introduced into transport by operator splitting. The dual porosity model use analytic solution and the non-linear adsorption is solved numerically by the Newton method.

The program is implemented in C/C++ using essentially PETSC library for linear algebra. The water flow as well as the transport simulation can be computed in parallel using MPI environment. This version also support output into VTK format, which is widely supported. In particular we recommend Paraview for visualization and postprocessing of the results.

The program is distributed under GNU GPL v. 3 licence and is available on the project web page: <http://dev.nti.tul.cz/trac/flow123d>

## Usage

On the Linux system the program can be started either directly or through a script `run_flow.sh`. When started directly by command

```
> flow123d -s example.ini
```

the program accepts one argument after swith `-s` which is the name of the principila input file. When you want to start a parallel job you shloud rather use starting script. Basic usage is:

```
> run_flow.sh -np 2 -s example.ini
```

which run simulation on 2 processes using the same INI file as before. For other possible arguments see the begining of the script.

On the Windows system you can start a squential run by command:

```
> flow123d.exe -s example.ini
```

or a parallel run by command:

```
> mpiexec.exe -np 2 flow123d.exe -s example.ini
```

The principal input file of the program is an INI file which contains names of other necessary input files. Those are the file with calculation mesh (**\*.msh**), the file with specification of neighbourings between dimensions (**\*.ngh**), the file with material description (**\*.mtr**) and the file with boundary conditions for the water flow problem (**\*.bcd**).

In the case of transport simulation one have to specify also the file with transport boundary conditions (**\*.tbc**) and the file with transport initial condition for individual substances (**\*.tic**).

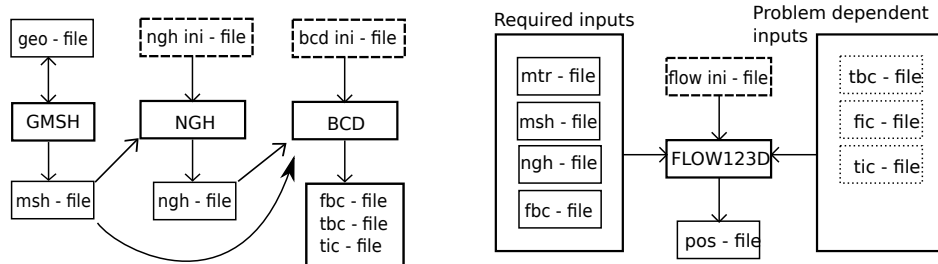


Figure 1: Preparation of input files.

For the preparation of input files we use several utilities (see Figure 1). We usually begin with a **\*.geo** file as a description of the domain geomery. This come as an input for the GMSH mesh generator, which produce the mesh file. Then we run program **ngh** to produce file of neighbourings. Finally we can use program **bcd** for the preparation of files with boundary and initial conditions. The file with material properties has to be created manualy, preferably by modifying some of the example problems. The programs **ngh** and **bcd** are distributed together with flow123d with their own limited documentation.

The output files can be either **\*.pos** files accepted by the GMSH or one can use VTK format that can be postprocessed by Paraview.

In following sections we briefly describe structure of individual input files.

## Flow123D ini file format

Section: **[Global]**

KEY	TYPE	DEFAULT	DESCRIPTION
<code>Problem_type</code>	<b>int</b>	NULL	Type of solved problem. Currently supported: 1 = steady saturated flow
<code>Description</code>	<b>string</b>	<i>undefined</i>	Short description of solved problem - any text.
<code>Stop_time</code>	<b>double</b>	1.0	Time interval of the whole problem.[time units]
<code>Save_step</code>	<b>double</b>	1.0	The output with transport is written every <code>Save_step</code> . [time units]

Section: **[Input]**

KEY	TYPE	DEFAULT	DESCRIPTION
<code>Mesh</code>	<b>string</b>	NULL	Name of file containig definition of the mesh for the problem.
<code>Material</code>	<b>string</b>	NULL	Name of file with hydraulical properties of the elements.
<code>Boundary</code>	<b>string</b>	NULL	Name of file with boundary condition data.
<code>Neighbouring</code>	<b>string</b>	NULL	Name of file describing topology of the mesh.
<code>Sources</code>	<b>string</b>	NULL	Name of file with definition of fluid sources. This is optional file, if this key is not defined, calculation goes on without sources.

Section: **[Transport]**

KEY	TYPE	DEFAULT	DESCRIPTION
Transport_on	<b>YES/NO</b>	NO	If set "YES" program compute transport too.
Sorption	<b>YES/NO</b>	NO	If set "YES" program include sorption too.
Dual_porosity	<b>YES/NO</b>	NO	If set "YES" program include dual porosity too.
Reactions	<b>YES/NO</b>	NO	If set "YES" program include reactions too.
Concentration	<b>string</b>	NULL	Name of file with initial concentration.
Transport_BCD	<b>string</b>	NULL	Name of file with boundary condition for transport.
Transport_out	<b>string</b>	NULL	Name of transport output file.
Transport_out_im	<b>string</b>	NULL	Name of transport immobile output file.
Transport_out_sorp	<b>string</b>	NULL	Name of transport sorbed output file.
Transport_out_im_sorp	<b>string</b>	NULL	Name of transport sorbed immobile output file.
N_substances	<b>int</b>	-1	Number of substances.
Substances	<b>string</b>	<i>undefined</i>	Names of the substances separated by commas.
Substances_density_scales	<b>list of doubles</b>	1.0	Scales of substances for the density flow calculation.

Section: **[Constants]**

KEY	TYPE	DEFAULT	DESCRIPTION
g	<b>double</b>	1.0	Gravity acceleration.
rho	<b>double</b>	1.0	Density of fluid.

Section: **[Run]**

KEY	TYPE	DEFAULT	DESCRIPTION
Screen_verbosity	<b>int</b>	8	Amount of messages printed on the screen. (0 = no messages, ..., 7 = all messages)
Pause_after_run	<b>YES/NO</b>	NO	If set to "YES", the program waits for a key press before it finishes.

Section: **[Solver]**

KEY	TYPE	DEFAULT	DESCRIPTION
<code>Use_last_solution</code>	<b>YES/NO</b>	NO	If set to "YES", uses last known solution for chosen solver.
<code>Solver_name</code>	<b>string</b>	petsc	Type of linear solver. Supported solvers are: <code>petsc</code> , <code>petsc.matis</code> (experimental)
<code>Solver_params</code>	<b>string</b>	NULL	PETSc options to override default choice of iterative solver and preconditioner (use with care). In particular to use UMFPACK sequential direct solver set: <code>Solve_params = "-ksp preonly -pc_type lu -pc_factor_mat_solver_package umfpack"</code> To use parallel direct solver MUMPS use: <code>Solve_params = "-ksp preonly -pc_type lu -pc_factor_mat_solver_package mumps -mat_mumps_icntl_14 5"</code>
<code>Keep_solver_files</code>	<b>YES/NO</b>	NO	If set to "YES", files for solver are not deleted after the run of the solver.
<code>Manual_solver_run</code>	<b>YES/NO</b>	NO	If set to "YES", programm stops after writing input files for solver and lets user to run it.
<code>Use_control_file</code>	<b>YES/NO</b>	NO	If set to "YES", programm do not create control file for solver, it uses given file.
<code>Control_file</code>	<b>string</b>	NULL	Name of control file for situation, when <code>Use_control_file</code> YES.
<code>NSchurs</code>	<b>int</b>	2	Number of Schur complements to use. Valid values are 0,1,2. The last one should be the fastest.
<code>Solver_accuracy</code>	<b>double</b>	1e-6	When to stop solver run - value of residum of matrix. Useful values from 1e-4 to 1e-10. Bigger number = faster run, less accuracy.
<code>max_it</code>	<b>int</b>	200	Maximum number of iteration of linear solver.

Section: **[Output]**

KEY	TYPE	DEFAULT	DESCRIPTION
Write_output_file	<b>YES/NO</b>	NO	If set to "YES", writes output file.
Output_file	<b>string</b>	NULL	Name of the output file (type 1).
Output_file_2	<b>string</b>	NULL	Name of the output file (type 2).
Output_digits	<b>int</b>	6	Number of digits used for floating point numbers in output file.
Output_file_type	<b>int</b>	1	Type of output file 1 - GMSH like format 2 - Flow data file 3 - both files (two separate names)
POS_view_params	<b>double[8]</b>	0 0 0 1 1 1 0 0	[x y z] angle of rotation "RotationX" [x y z] scaling "ScaleX" [x y] screen position shift "TranslationX"
Pos_format	<b>string</b>	ASCII	Output file format. One can use: ASCII, BIN, or VTK_SERIAL_ASCII

Description: Options controlling output file of the programm

## Section: [Semchem\_module]

KEY	TYPE	DEFAULT	DESCRIPTION
Compute_reactions	Yes/No	"No"	NO = transport without chemical reactions YES = transport influenced by chemical reactions
Output_precision	int	1	Number of decimal places written to output file created by Semchem_module.
Number_of_further_species	int	0	Concentrations of these species are not computed, because they are ment to be unexhaustible.
Temperature	double	0.0	Temperature, one of state variables of the system.
Temperature_Gf	double	0.0	Temperature at which Free Gibbs Energy is specified.
Param_Afi	double	0.0	Parameter of the Debye-Hückel equation for activity coefficients computation.
Param_b	double	0.0	Parameter of the Debye-Hückel equation for activity coefficients computation.
Epsilon	double	0.0	Epsilon specifies relative norm of residuum estimate to stop numerical algorithms used by Semchem_module.
Time_steps	int	1	Number of transport step subdivisions for Semchem_module.
Slow_kinetics_substeps	int	0	Number of substeps performed by Runge-Kutta method used for slow kinetics simulation.
Error_norm_type	string	"Absolute"	Through wich kind of norm the error is measured.
Scalling	boolean	"No"	Type of the problem preconditioning for better convergence of numerical method.



Section: **[Aqueous\_species]**

KEY	TYPE	DEFAULT	DESCRIPTION
El_charge	<b>int</b>	0	Electric charge of an Aqueous_specie particle under consideration.
dGf	<b>double</b>	0.0	Free Gibbs Energy valid for TemperatureGf.
dHf	<b>double</b>	0.0	Enthalpy
Molar_mass	<b>double</b>	0.0	Molar mass of Aqueous_species.

Section: **[Further\_species]**

KEY	TYPE	DEFAULT	DESCRIPTION
Specie_name	<b>string</b>	""	Name belonging to Further_specie under consideration.
dGf	<b>double</b>	0.0	Free Gibbs Energy valid for TemperatureGf.
dHf	<b>double</b>	0.0	Enthalpy
Molar_mass	<b>double</b>	0.0	Molar mass of Further_species.
Activity	<b>double</b>	0.0	Activity of Further_species.

Section: **[Reaction\_i]**

KEY	TYPE	DEFAULT	DESCRIPTION
Reaction_type	<b>string</b>	"unknown"	Type of considered reaction (Equilibrium, Kinetics, Slow_kinetics).
Stoichiometry	<b>int</b>	0	Stoichiometric coefficients of species taking part in i-th reaction.
Kinetic_constant	<b>double</b>	0.0	Kinetic constant for determination of reaction rate.
Order_of_reaction	<b>int</b>	0	Order of kinetic reaction for participating species.
Equilibrium_constant	<b>double</b>	0.0	Equilibrium constant defining i-th reaction.

# Mesh file format version 2.0

The mesh file format comes from the GMSH system. Following text is copied from the GMSH documentation.

===== BEGIN OF INSERTED TEXT =====

Version 2.0 of the .MSH file format is Gmsh's new native mesh file format. It is very similar to the old one (Version 1.0), but is more general: it contains information about itself and allows to associate an arbitrary number of integer tags with each element.

The .MSH file format, version 2.0, is divided in three sections, defining the file format (\$MeshFormat-\$EndMeshFormat), the nodes (\$Nodes-\$EndNodes) and the elements (\$Elements-\$EndElements) in the mesh:

```
$MeshFormat
2.0 file-type data-size
$EndMeshFormat
$Nodes
number-of-nodes
node-number x-coord y-coord z-coord
...
$EndNodes
$Elements
number-of-elements
elm-number elm-type number-of-tags <tags> node-number-list
...
$EndElements
```

where:

*file-type* is an integer equal to 0 in the ASCII file format.

*data-size* is an integer equal to the size of the floating point numbers used in the file (usually, *data-size* = sizeof(double)).

*number-of-nodes* is the number of nodes in the mesh.

*node-number* is the number (index) of the *n*-th node in the mesh. Note that the *node-numbers* do not have to be given in a consecutive (or even an ordered) way.

*x-coord y-coord z-coord* are the floating point values giving the X, Y and Z coordinates of the *n*-th node.

*number-of-elements* is the number of elements in the mesh.

*elm-number* is the number (index) of the *n*-th element in the mesh. Note that the *elm-numbers* do not have to be given in a consecutive (or even an ordered) way.

*elm-type* defines the geometrical type of the *n*-th element:

- 1 Line (2 nodes)
- 2 Triangle (3 nodes)
- 3 Quadrangle (4 nodes)
- 4 Tetrahedron (4 nodes)
- 5 Hexahedron (8 nodes)
- 6 Prism (6 nodes)
- 7 Pyramid (5 nodes)
- 8 Second order line (3 nodes)
- 9 Second order triangle (6 nodes)
- 11 Second order tetrahedron (10 nodes)
- 15 Point (1 node)

*number-of-tags* gives the number of tags for the *n*-th element. By default, Gmsh generates meshes with two tags and reads files with an arbitrary number of tags: see below.

*tag* is an integer tag associated with the *n*-th element. By default, the first tag is the number of the physical entity to which the element belongs; the second is the number of the elementary geometrical entity to which the element belongs; the third is the number of a mesh partition to which the element belongs.

*node-number-list* is the list of the node numbers of the *n*-th element (separated by white space, without commas). The ordering of the nodes is given in Gmsh node ordering; for second order elements, the first order nodes are given first, followed by the nodes associated with the edges, followed by the nodes associated with the faces (if any). The ordering of these additional nodes follows the ordering of the edges/faces given in Gmsh node ordering.

===== END OF INSERTED TEXT =====

More information about GMSH can be found at its homepage:  
<http://www.geuz.org/gmsh/>

## Comments concerning 1-2-3-FLOW:

- Every inconsistency of the file stops the calculation. These are:
  - Existence of nodes with the same *node-number*.
  - Existence of elements with the same *elm-number*.
  - Reference to non-existing node.
  - Reference to non-existing material (see below).
  - Difference between *number-of-nodes* and actual number of lines in nodes' section.
  - Difference between *number-of-elements* and actual number of lines in elements' section.
- By default 1-2-3-FLOW uses meshes with *number-of-tags* = 2.

*tag1* is number of region in which the element lies.

*tag2* is number of material (reference to .MTR file) in the element.

- Currently, line ( $type = 1$ ), triangle ( $type = 2$ ) and tetrahedron ( $type = 4$ ) are the only supported types of elements. Existence of an element of different type stops the calculation.
- Wherever possible, we use the file extension `.MSH`. It is not required, but highly recommended.

# Material properties file format, version 1.0

The file is divided in two sections, header and data. The extension `.MTR` is highly recommended for files of this type.

```
$MaterialFormat
1.0 file-type data-size
$EndMaterialFormat
$ConductivityTensor
number-of-materials
material-number tensor-type <tensor-type-specific-data> [text]
...
$EndConductivityTensor
$Storativity
material-number <storativity-coefficient> [text]
...
$EndStorativity
$Geometry
material-number geometry-type <geometry-type-specific-coefficient> [text]
...
$EndGeometry
$Sorption
material-number substance-id sorption-type <sorption-type-specific-data> [text]
...
$EndSorption
$SorptionFraction
material-number <sorption-fraction-coefficient> [text]
...
$EndSorptionFraction
$DualPorosity
material-number <mobile-porosity-coefficient> <immobile-porosity-coefficient>
<nonequilibrium-coefficient-substance(0)> ...<nonequilibrium-coefficient-substance(n-1)>
[text]
...
$EndDualPorosity
$Reactions
reaction-type <reaction-type-specific-coefficient> [text]
...
$EndReactions
```

where:

*file-type* **int** — is equal 0 for the ASCII file format.

*data-size* **int** — the size of the floating point numbers used in the file. Usually *data-size* = sizeof(double).

*number-of-materials* **int** — Number of materials defined in the file.

*material-number* **int** — is the number (index) of the n-th material. These numbers do not have to be given in a consecutive (or even an ordered) way. Each number has to be

given only once, multiple definition are treated as inconsistency of the file and cause stopping the calculation (exception \$Sorption section).

*tensor-type* **int** — is type of the material, see table.

*<tensor-type-specific-data >* — format of this list depends on the *tensor-type*.

*<storativity-coefficient>* **double** — coefficient of storativity

*geometry-type* **int** — type of complement dimension parameter (only for 1D and 2D material), for 1D element is supported type 1 - cross-section area, for 2D element is supported type 2 - thickness.

*<geometry-type-specific-coefficient>* **double** — cross-section for 1D element or thickness for 2D element.

*substance-id* **int** — refers to number of transported substance, numbering starts on 0.

*sorption-type* **int** — type 1 - linear sorption isotherm, type 2 - Freundlich sorption isotherm, type 3 - Langmuir sorption isotherm.

*<sorption-type-specific-data >* — format of this list depends on the *sorption - type*, see table.

Note: Section \$Sorption is needed for calculation only if *Sorption* is turned on in the *ini* file.

*<sorption-fraction-coefficient>* **double** — ratio of the "mobile" solid surface in the contact with "mobile" water to the total solid surface (this parameter (section) is needed for calculation only if *Dual\_porosity* and *Sorption* is together turned on in the ini file).

*<mobile-porosity-coefficient>* **double** — ratio of the mobile pore volume to the total volume (this parameter is needed only if *Transport\_on* is turned on in the ini file).

*<immobile-porosity-coefficient>* **double** — ratio of the immobile pore volume to the total pore volume (this parameter is needed only if *Dual\_porosity* is turned on in the ini file).

*<nonequilibrium-coefficient-substance(i)>* **double** — nonequilibrium coefficient for substance  $i$ ,  $\forall i \in \langle 0, n - 1 \rangle$  where  $n$  is number of transported substances (this parameter is needed only if *Dual\_porosity* is turned on in the ini file).

*reaction-type* **int** — type 0 - zero order reaction

*<reaction-type-specific-data >* — format of this list depends on the *reaction - type*, see table.

<i>tensor-type</i>	<i>tensor-type-specific-data</i>	Description
11	$k$	$\mathbf{K} = (k)$
-11	$a$	$\mathbf{A} = \mathbf{K}^{-1} = (a)$
21	$k$	$\mathbf{K} = \begin{pmatrix} k & 0 \\ 0 & k \end{pmatrix}$
22	$k_x \quad k_y$	$\mathbf{K} = \begin{pmatrix} k_x & 0 \\ 0 & k_y \end{pmatrix}$
23	$k_x \quad k_y \quad k_{xy}$	$\mathbf{K} = \begin{pmatrix} k_x & k_{xy} \\ k_{xy} & k_y \end{pmatrix}$
-21	$a$	$\mathbf{A} = \mathbf{K}^{-1} = \begin{pmatrix} a & 0 \\ 0 & a \end{pmatrix}$
-22	$a_x \quad a_y$	$\mathbf{A} = \mathbf{K}^{-1} = \begin{pmatrix} a_x & 0 \\ 0 & a_y \end{pmatrix}$
-23	$a_x \quad a_y \quad a_{xy}$	$\mathbf{A} = \mathbf{K}^{-1} = \begin{pmatrix} a_x & a_{xy} \\ a_{xy} & a_y \end{pmatrix}$
31	$k$	$\mathbf{K} = \begin{pmatrix} k & 0 & 0 \\ 0 & k & 0 \\ 0 & 0 & k \end{pmatrix}$
33	$k_x \quad k_y \quad k_z$	$\mathbf{K} = \begin{pmatrix} k_x & 0 & 0 \\ 0 & k_y & 0 \\ 0 & 0 & k_z \end{pmatrix}$
36	$k_x \quad k_y \quad k_z \quad k_{xy} \quad k_{xz} \quad k_{yz}$	$\mathbf{K} = \begin{pmatrix} k_x & k_{xy} & k_{xz} \\ k_{xy} & k_y & k_{yz} \\ k_{xz} & k_{yz} & k_z \end{pmatrix}$
-31	$a$	$\mathbf{A} = \mathbf{K}^{-1} = \begin{pmatrix} a & 0 & 0 \\ 0 & a & 0 \\ 0 & 0 & a \end{pmatrix}$
-33	$a_x \quad a_y \quad a_z$	$\mathbf{A} = \mathbf{K}^{-1} = \begin{pmatrix} a_x & 0 & 0 \\ 0 & a_y & 0 \\ 0 & 0 & a_z \end{pmatrix}$
-36	$a_x \quad a_y \quad a_z \quad a_{xy} \quad a_{xz} \quad a_{yz}$	$\mathbf{A} = \mathbf{K}^{-1} = \begin{pmatrix} a_x & a_{xy} & a_{xz} \\ a_{xy} & a_y & a_{yz} \\ a_{xz} & a_{yz} & a_z \end{pmatrix}$

Note: all variables (  $k$ ,  $k_x$ ,  $k_y$ ,  $k_z$ ,  $k_{xy}$ ,  $k_{xz}$ ,  $k_{yz}$ ,  $a$ ,  $a_x$ ,  $a_y$ ,  $a_z$ ,  $a_{xy}$ ,  $a_{xz}$ ,  $a_{yz}$  ) are of the double type.

<i>sorption-type</i>	<i>sorption-type-specific-data</i>	Description
1	$k_D[1]$	$s = k_D c$
2	$k_F[(L^{-3} \cdot M^1)^{(1-\alpha)}] \quad \alpha[1]$	$s = k_F c^\alpha$
3	$K_L[L^3 \cdot M^{-1}] \quad s^{max}[L^{-3} \cdot M^1]$	$s = \frac{K_L s^{max} c}{1 + K_L c}$

Note: all variables (  $k_D$ ,  $k_F$ ,  $\alpha$ ,  $K_L$ ,  $s^{max}$  ) are of the double type.

<i>reaction-type</i>	<i>reaction-type-specific-data</i>	Description
0	$substance-id[1] \quad k[M \cdot L^{-3} \cdot T^{-1}]$	$\frac{\partial c_m^{[substance-id]}}{\partial t} = k$

Where  $c_m^{[substance-id]}$  is mobile concentration of substance with id *substance-id* and  $\Delta t$  is the internal transport time step defined by CFL condition.

*text* **char**[] — is a text description of the material, up to 256 chars. This parameter is

optional.

**Comments concerning 1-2-3-FLOW:**

- If *number-of-materials* differs from actual number of material lines in the file, it stops the calculation.



# Boundary conditions file format, version 1.0

The file is divided in two sections, header and data.

```
$BoundaryFormat
1.0 file-type data-size
$EndBoundaryFormat
$BoundaryConditions
number-of-conditions
condition-number type <type-specific-data> where <where-data> number-of-tags <tags>
[text]
...
$EndBoundaryConditions
```

where

*file-type* **int** — is equal 0 for the ASCII file format.

*data-size* **int** — the size of the floating point numbers used in the file. Usually *data-size* = sizeof(double).

*number-of-conditions* **int** — Number of boundary conditions defined in the file.

*condition-number* **int** — is the number (index) of the n-th boundary condition. These numbers do not have to be given in a consecutive (or even an ordered) way. Each number has to be given only once, multiple definition are treated as inconsistency of the file and cause stopping the calculation.

*type* **int** — is type of the boundary condition. See below for definitions of the types.

<*type-specific-data*> — format of this list depends on the *type*. See below for specification of the *type-specific-data* for particular types of the boundary conditions.

*where* **int** — defines the way, how the place for the condition is prescribed. See below for details.

<*where-data*> — format of this list depends on *where* and actually defines the place for the condition. See below for details.

*number-of-tags* **int** — number of integer tags of the boundary condition. It can be zero.

< *tags* > *number-of-tags*\***int** — list of tags of the boundary condition. Values are separated by spaces or tabs. By default we set *number-of-tags*=1, where *tag1* defines group of boundary conditions, "type of water" in our jargon. This can be used to calculate total fluxes through the boundary group.

[*text*] **char**[] — arbitrary text, description of the fracture, notes, etc., up to 256 chars. This is an optional parameter.

## Types of boundary conditions and their data

*type* = 1 — Boundary condition of the Dirichlet's type

*type* = 2 — Boundary condition of the Neumann's type

*type* = 3 — Boundary condition of the Newton's type

<i>type</i>	<i>type-specific-data</i>	Description
1	<i>scalar</i>	Prescribed value of pressure or piez. head
2	<i>flux</i>	Prescribed value of flux through the boundary
3	<i>scalar sigma</i>	Scalar value and the $\sigma$ coefficient

*scalar*, *flux* and *sigma* are of the `double` type.

## Ways of defining the place for the boundary condition

*where* = 1 — Condition on a node

*where* = 2 — Condition on a (generalized) side

*where* = 3 — Condition on side for element with only one external side.

<i>where</i>	<i>&lt;where-data&gt;</i>	Description
1	<i>node-id</i>	Node id number, according to <code>.MSH</code> file
2	<i>elm-id sid-id</i>	Elm. id number, local number of side
3	<i>elm-id</i>	Elm. id number

The variables *node-id*, *elm-id*, *sid-id* are of the `int` type.

## Comments concerning 1-2-3-FLOW:

- We assume homegemous Neumman's condition as the default one. Therefore we do not need to prescribe conditions on the whole boundary.
- If the condition is given on the inner edge, it is treated as an error and stops calculation.
- Any inconsistence in the file stops calculation. (Bad number of conditions, multiple definition of condition, reference to non-existing node, etc.)
- At least one of the conditions has to be of the Dirichlet's or Newton's type. This is well-known fact from the theory of the PDE's.
- Local numbers of sides for *where* = 2 must be lower than the number of sides of the particular element and greater then or equal to zero.
- The element specified for *where* = 3 must have only one external side, otherwise the program stops.

# Neighbouring file format, version 1.0

The file is divided in two sections, header and data. The extension **.NGH** is highly recommended for files of this type.

```
$NeighbourFormat
1.0 file-type data-size
$EndNeighbourFormat
$Neighbours
number-of-neighbours
neighbour-number type <type-specific-data>
...
$EndNeighbours
```

where

*file-type* **int** — is equal 0 for the ASCII file format.

*data-size* **int** — the size of the floating point numbers used in the file. Usually *data-size* = sizeof(double).

*number-of-neighbours* **int** — Number of neighbouring defined in the file.

*neighbour-number* **int** — is the number (index) of the n-th neighbouring. These numbers do not have to be given in a consecutive (or even an ordered) way. Each number has to be given only once, multiple definition are treated as inconsistency of the file and cause stopping the calculation.

*type* **int** — is type of the neighbouring.

*<type-specific-data>* — format of this list depends on the *type*.

## Types of neighbouring and their specific data

*type* = 10 — “Edge with common nodes”, i.e. sides of elements with common nodes. (Possible many elements)

*type* = 11 — “Edge with specified sides”, i.e. sides of the edge are explicitly defined. (Possible many elements)

*type* = 20 — “Compatible”, i.e. volume of an element with a side of another element. (Only two elements)

*type* = 30 — “Non-compatible” i.e. volume of an element with volume of another element. (Only two elements)

<i>type</i>	<i>type-specific-data</i>	Description
10	<i>n_elm eid1 eid2 ...</i>	number of elements and their ids
11	<i>n_sid eid1 sid1 eid2 sid2 ...</i>	number of sides, their elements and local ids
20	<i>eid1 eid2 sid2 coef</i>	Elm 1 has to have lower dimension
30	<i>eid1 eid2 coef</i>	Elm 1 has to have lower dimension

*coef* is of the **double** type, other variables are **ints**.

## Comments concerning 1-2-3-FLOW:

- Every inconsistency or error in the .NGH file causes stopping the calculation. These are especially:
  - Multiple usage of the same *neighbour-number*.
  - Difference between *number-of-neighbours* and actual number of data lines.
  - Reference to nonexistent element.
  - Nonsense number of side.
- The variables *sid?* must be nonnegative and lower than the number of sides of the particular element.

## Sources file format, version 1.0

The file is divided in two sections, header and data. The extension `.SRC` is highly recommended for files of this type.

```
$SourceFormat
1.0 file-type data-size
$EndSourceFormat
$Sources
number-of-sources
eid density
...
$EndSources
```

where

*file-type* **int** — is equal 0 for the ASCII file format.

*data-size* **int** — the size of the floating point numbers used in the file. Usually *data-size* = `sizeof(double)`.

*number-of-sources* **int** — Number of sources defined in the file.

*eid* **int** — is id-number of the element, where the source lies.

*density* **double** — is the density of the source, in volume of fluid per time unit. Positive values are sources, negative are sinks.

### Comments concerning 1-2-3-FLOW:

- Every inconsistency or error in the `.SRC` file causes stopping the calculation. These are especially:
  - Multiple usage of the same *source-number*.
  - Difference between *number-of-sources* and actual number of data lines.
  - Reference to nonexisting element.

# ASCII post-processing file format version 1.2

File format of this file comes from the GMSH system. Following text is copied from the GMSH documentation.

===== BEGIN OF INSERTED TEXT =====

The ASCII post-processing file is divided in several sections: one format section, enclosed between `$PostFormat-$EndPostFormat` tags, and one or more post-processing views, enclosed between `$View-$EndView` tags:

`$PostFormat`

`1.2 file-type data-size`

`$EndPostFormat`

`$View`

`view-name nb-time-steps`

`nb-scalar-points nb-vector-points nb-tensor-points`

`nb-scalar-lines nb-vector-lines nb-tensor-lines`

`nb-scalar-triangles nb-vector-triangles nb-tensor-triangles`

`nb-scalar-quadrangles nb-vector-quadrangles nb-tensor-quadrangles`

`nb-scalar-tetrahedra nb-vector-tetrahedra nb-tensor-tetrahedra`

`nb-scalar-hexahedra nb-vector-hexahedra nb-tensor-hexahedra`

`nb-scalar-prisms nb-vector-prisms nb-tensor-prisms`

`nb-scalar-pyramids nb-vector-pyramids nb-tensor-pyramids`

`nb-text2d nb-text2d-chars nb-text3d nb-text3d-chars`

`<time-step-values>`

`<scalar-point-values>`

`<vector-point-values>`

`<tensor-point-values>`

`<scalar-line-values>`

`<vector-line-values>`

`<tensor-line-values>`

`<scalar-triangle-values>`

`<vector-triangle-values>`

`<tensor-triangle-values>`

`<scalar-quadrangle-values>`

`<vector-quadrangle-values>`

`<tensor-quadrangle-values>`

`<scalar-tetrahedron-values>`

`<vector-tetrahedron-values>`

`<tensor-tetrahedron-values>`

`<scalar-hexahedron-values>`

`<vector-hexahedron-values>`

`<tensor-hexahedron-values>`

`<scalar-prism-values>`

`<vector-prism-values>`

`<tensor-prism-values>`

`<scalar-pyramid-values>`

```

<vector-pyramid-values>
<tensor-pyramid-values>
<text2d> <text2d-chars>
<text3d> <text3d-chars>
$EndView

```

where:

*file-type* is an integer equal to 0 in the ASCII file format.

*data-size* is an integer equal to the size of the floating point numbers used in the file (usually, *data-size* = sizeof(double)).

*view-name* is a string containing the name of the view (max. 256 characters).

*nb-time-steps* is an integer giving the number of time steps in the view.

*nb-scalar-points*, *nb-vector-points*, ... are integers giving the number of scalar points, vector points, ... in the view.

*nb-text2d*, *nb-text3d* are integers giving the number of 2D and 3D text strings in the view.

*nb-text2d-chars*, *nb-text3d-chars* are integers giving the total number of characters in the 2D and 3D strings.

*time-step-values* is a list of *nb-time-steps* double precision numbers giving the value of the time (or any other variable) for which an evolution was saved.

*scalar-point-value*, *vector-point-value*, ... are lists of double precision numbers giving the node coordinates and the values associated with the nodes of the *nb-scalar-points* scalar points, *nb-vector-points* vector points, ..., for each of the *time-step-values*.

For example, *vector-triangle-value* is defined as:

```

coord1-node1 coord1-node2 coord1-node3
coord2-node1 coord2-node2 coord2-node3
coord3-node1 coord3-node2 coord3-node3
comp1-node1-time1 comp2-node1-time1 comp3-node1-time1
comp1-node2-time1 comp2-node2-time1 comp3-node2-time1
comp1-node3-time1 comp2-node3-time1 comp3-node3-time1
comp1-node1-time2 comp2-node1-time2 comp3-node1-time2
comp1-node2-time2 comp2-node2-time2 comp3-node2-time2
comp1-node3-time2 comp2-node3-time2 comp3-node3-time2
...

```

*text2d* is a list of 4 double precision numbers:

```
coord1 coord2 style index
```

where *coord1* and *coord2* give the coordinates of the leftmost element of the 2D string in screen coordinates, *index* gives the starting index of the string in *text2d-chars* and *style* is currently unused.

*text2d-chars* is a list of *nb-text2d-chars* characters. Substrings are separated with the ‘^’ character (which is a forbidden character in regular strings).

*text3d* is a list of 5 double precision numbers

*coord1 coord2 coord3 style index*

where *coord1*, *coord2* and *coord3* give the coordinates of the leftmost element of the 3D string in model (real world) coordinates, *index* gives the starting index of the string in *text3d-chars* and *style* is currently unused.

*text3d-chars* is a list of *nb-text3d-chars* chars. Substrings are separated with the ‘^’ character.

===== END OF INSERTED TEXT =====

More information about GMSH can be found at its homepage:  
<http://www.geuz.org/gmsh/>

## Comments concerning FFL0W20:

- FFL0W20 generates .POS file with four views: Elements’ pressure, edges’ pressure, interelement fluxes and complex view. First three views shows ”raw data”, results obtained by the solver without any interpolations, smoothing etc. The fourth view contains data processed in this way.

**Elements’ pressure:** Contains only *scalar-triangle-values*. Triangles are the same as the elements of the original mesh. We prescribe constant value of the pressure on the element, as it was calculated by the solver as the unknown  $p$ . Therefore, the three values on every triangle are the same.

**Edge pressure:** Contains only *scalar-line-values*. The lines are the same as the edges of the elements of the original mesh. We prescribe constant value of the pressure on the edge, as it was calculated by the solver as the unknown  $\lambda$ . Therefore, the two values on every edge are the same.

**Interelement flux:** Contains *vector-point-values* and *scalar-triangle-values*. The *scalar-triangle-values* carry no information, all values are set to 0, these are in the file only to define a shape of the elements. The points for the *vector-point-values* are midpoints of the sides of the elements. The vectors are calculated as  $u\mathbf{n}$ , where  $u$  is value of the flux calculated by the solver and  $\mathbf{n}$  is normalized vector of outer normal of the element’s side.

**Complex view:** Contains *scalar-triangle-values* and *vector-point-values*. The *scalar-triangle-values* shows the shape of the pressure field. The triangles are the the same as the elements of the original mesh. Values of pressure in nodes are interpolated from  $p_s$  and  $\lambda_s$ . The *vector-point-values* shows the velocity of the flow in the centres of the elements.