

Hydrothermal modeling of a deep geothermal reservoir including a hydraulic fracture

Step by Step Tutorial

Norihiro Watanabe^{1, 2}, Herbert Kunz³, Olaf Kolditz^{1, 2}

1) Helmholtz Centre for Environmental Research (UFZ)

2) Technical University of Dresden (TUD)

3) Federal Institute for Geosciences and Natural Resources (BGR)

OpenGeoSys – Preprint
Version 0.2 [2012-06]

Leipzig, June 2012

Table of Contents

1	Introduction	3
1.1	OpenGeoSys.....	3
1.2	Prerequisite for Windows users	3
2	Theory	5
2.1	Hydrothermal processes in discrete fracture-porous medium systems...	5
2.2	References	6
3	Example.....	7
4	Pre-processing	10
4.1	Start-up GINA_OGS	10
4.2	Meshing (*.msh)	11
4.3	Geometric object definition (*.gli)	14
4.4	Process definition (*.pcs)	14
4.5	Initial and boundary conditions	15
4.5.1	Initial conditions (*.ic)	15
4.5.2	Dirichlet boundary conditions (*.bc)	18
4.5.3	Source Terms inc. Neumann BC (*.st)	18
4.6	Material properties	19
4.6.1	Fluid phase properties (*.mfp)	19
4.6.2	Solid phase properties (*.msp)	20
4.6.3	Medium properties (*.mmp)	21
4.7	Time stepping (*.tim)	22
4.8	Other settings.....	23
4.8.1	Numerical properties (*.num)	23
4.8.2	Output Parameters (*.out)	24
5	Simulation.....	27
6	Post-processing with ParaView	28
6.1	Load PVD/VTU files.....	28
6.2	Plot temperature isosurface	31
6.3	Plot only fracture elements.....	32
6.4	Plot flow vector in fracture elements.....	34
7	Advanced topics	35
7.1	Variable fluid properties	35
7.2	Reducing numerical oscillations in heat transport.....	37
7.3	Output elemental velocity.....	39
7.4	Steady-state modeling	39
7.5	Conservative tracer transport modeling.....	39
7.6	Time-dependent boundary condition	42
7.7	Importing/exporting simulation results as initial conditions	42

1 Introduction

This step-by-step tutorial is prepared to help OpenGeoSys users to simulate hydrothermal processes in deep geothermal reservoirs including hydraulic fractures. We begin with a brief introduction of OpenGeoSys and required software. Next, theoretical background of simulation techniques is shortly explained in Section 2. Description of an example used in this tutorial is given in Section 3. In Section 4, 5 and 6, step-by-step procedures are presented for pre-processing, simulations and post-processing to solve the given example.

1.1 OpenGeoSys

OpenGeoSys (OGS) is an open scientific source code for simulation of thermo-hydro-mechanical-chemical (THMC) processes in porous-fractured media with open applications in groundwater and surface hydrology, water resources, geotechnics, radioactive waste disposal, geothermics, and CO₂ storage (Wang & Kolditz, 2007).

OGS has been used to simulate THM processes in deep geothermal reservoirs, e.g. in Rosemanowes in UK (Kolditz & Clauser, 1998), Soultz-sous-Forêts in the Rhine Valley (Kolditz, 2001), Urach Spa in the Swabian Alb (McDermott et al. 2006).

1.2 Prerequisite for Windows users

Although OpenGeoSys is a cross-platform application, this tutorial is prepared for Windows users because the following pre-processing steps rely on software which is only available on the platform.

Users should have the following software installed on their computers:

- OpenGeoSys: Simulation software
- GINA_OGS: Pre-Post processing software for OpenGeoSys
- ParaView: Open-source 3D Visualization tool

* For lectures, we prepare an installation CD which includes all necessary software. Insert the CD into your computer and copy files on your hard disk.

To run OpenGeoSys, the following files are necessary:

ogs.exe	Windows application
---------	---------------------

- You can download the executable from the OpenGeoSys website (www.opengeosys.net).
- OpenGeoSys is a cross platform application. Users can compile source codes on Linux or get precompiled files from the website.

GINA_OGS is an interactive graphical user interface for the FE-Code OpenGeoSys including geometry and mesh generation (also with GMSH), preparation of the FE-calculation and visualization of the calculated data. For the post-processing GINA_OGS provides an interface to TECPLOT. The software is developed by Herbert Kunz (BGR Hannover). For details, please refer its manual. The following files are necessary to run GINA_OGS:

Gina_OGS.exe	Windows application
--------------	---------------------

comdlg32.ocx, msflxgrd.ocx, richtx32.ocx, comctl32.ocx, mscomctl.ocx, msvbvm60.dll	System files (Visual Basic 6.0 Runtime library)
--	--

Users can plot simulation results on GINA_OGS. However, for large dataset, we recommend ParaView to visualize and analyze the results. ParaView is an open-source, multiplatform application that is used for viewing 2-D and 3-D data sets. ParaView has the capability to visualize very large data sets in parallel. ParaView supports a lot of file formats. For example, VTK, PLOT3D, DEM, STL, AVS files are available. ParaView is available from <http://www.paraview.org/>.

2 Theory

2.1 Hydrothermal processes in discrete fracture-porous medium systems

Physical system in deep geothermal reservoirs incorporates non-isothermal fluid flow in fractured porous media. In this tutorial, rock matrices are modeled as porous media, whereas hydraulic fractures are idealized as lower-dimensional hydraulic conduits, e.g. lines in two-dimensional space. This implies that fluid pressure or temperature is uniform across the fracture width. In addition, instantaneous exchange of flow and heat between porous medium and fracture systems are assumed. Local equilibrium of thermodynamics is assumed and macroscopic balance equations are considered to derive the governing equations. The corresponding field variables are liquid phase pressure p , temperature T . More details on THM mechanics can be found e.g. in Wang & Kolditz (2007).

Hydraulic flow in porous media can be described by using the mass balance equation for a unit volume as

$$S_s \frac{\partial p}{\partial t} - \nabla \cdot \left(\frac{\mathbf{k}}{\mu} (\nabla p + \rho^l \mathbf{g}) \right) = Q_f$$

where $S_s = (\alpha - n) / K_s + n / K_l$ is specific storage of the medium with α the Biot-Willis coefficient, n porosity, K_s the compressibility of solid and K_l the compressibility of liquid. p is liquid pressure, \mathbf{k} is intrinsic permeability tensor, μ is fluid dynamic viscosity, \mathbf{g} is gravity acceleration vector and Q_f is volumetric fluid source/sink term.

Heat transport equation taking into account advective and diffusive fluxes is

$$\overline{\rho c_p} \frac{\partial T}{\partial t} + \rho^l c_p^l \mathbf{v} \cdot \nabla T - \nabla \cdot (\bar{\lambda} \nabla T) = Q_T$$

where $\overline{\rho c_p} = n \rho^l c_p^l + (1-n) \rho^s c_p^s$ is heat storage of porous medium with n porosity, c_p^l specific heat capacity of fluid, ρ^l fluid density, c_p^s specific heat capacity of rock and ρ^s rock density. \mathbf{v} is Darcy velocity, Q_T heat source/sink term, $\bar{\lambda} = n \lambda^l + (1-n) \lambda^s$ heat conductivity of porous medium with λ^l the fluid, λ^s the rock heat conductivity.

In discrete fractures with an assumption of no infill materials, hydraulic flow can be described in its local coordinates from the mass balance law as

$$b S_s^f \frac{\partial p}{\partial t} - \bar{\nabla} \cdot \left(\frac{\mathbf{k}^f}{\mu} (\bar{\nabla} p + \rho^l \mathbf{g}) \right) = Q_f$$

where b is a fracture aperture, $S_s^f = \alpha / K_l$ is specific storage of the fracture. \mathbf{k}^f is fracture permeability which can be calculated by Cubic law, i.e. $k^f = b^2 / 12$. Heat transport equation in discrete fractures is

$$b \rho^l c_p^l \frac{\partial T}{\partial t} + b \rho^l c_p^l \mathbf{v} \cdot \bar{\nabla} T - \nabla \cdot (b \lambda^l \bar{\nabla} T) = Q_T$$

Remarks

- Unit of temperature is Kelvin.
- OGS (ver. 5.3.01) does not distinguish a mechanical and hydraulically ef-

fective aperture.

Table 1 List of symbols

Symbol	Parameter
b	Fracture aperture [m]
C_p	Specific heat [J/kg K]
\mathbf{g}	Gravity acceleration vector [m/s ²]
\mathbf{k}	Intrinsic permeability tensor [m ²]
K	Compressibility of solid and liquid phase
\mathbf{I}	Identity tensor
n	Porosity
P	Liquid phase pressure [Pa]
Q_T	Heat sour/sink [J/s] or [W]
Q_f	Flow source/sink [m ³ /s]
S_s	Specific storage [1/Pa]
t	Time [s]
T	Temperature [K]
\mathbf{v}	Darcy velocity [m/s]
α	the Biot-Willis coefficient
μ	dynamic viscosity [Pa s]
ρ	Density [kg/m ³]
λ	Thermal conductivity [W/m K]

2.2 References

More detail on geothermal reservoir simulation information is available in the following literatures.

- Kolditz, O. (1995a): Modelling of flow and heat transfer in fractured rock: Conceptual model of a 3-D deterministic fracture network. *Geothermics*, 24 (3): 451-470.
- Kolditz, O. (1995b): Modelling of flow and heat transfer in fractured rock: Dimensional effect of matrix heat diffusion. *Geothermics*, 24 (3): 421-438.
- Kolditz O and Clauser C (1998): Numerical simulation of flow and heat transfer in fractured crystalline rocks: Application to the hot dry rock site at Rosemanowes (UK). *Geothermics*, 28(1): 1-26.
- Kolditz O (2001): Non-linear flow in fractured rock. *Int J. Numerical Methods in Fluid and Heat Transport*, vol 11(6): 547-576.
- McDermott CI, Randriamanjatoa AL, Tenzer H and Kolditz O (2006): Simulation of heat extraction from crystalline rocks: The influence of coupled processes on differential reservoir cooling. *Geothermics*, vol. 35 (3): 321-344.
- McDermott, C. I., M. Lodemann, I. Ghergut, H. Tenzer, M. Sauter and O. Kolditz (2006): Investigation of Coupled Hydraulic-Geomechanical Processes – Field Experiments and Numerical Analysis at the KTB site. *Geofluids*, vol 6(1): 67-81.
- Wang W and Kolditz O (2007): Object-oriented finite element analysis of thermo-hydro-mechanical (THM) problems in porous media, *Int. J. Numerical Methods in Engineering*, vol. 69 (1): 162-201.
- Watanabe N, McDermott C, Wang W, Taniguchi T, Kolditz O (2009): Uncertainty analysis of thermo-hydro-mechanical processes in heterogeneous porous media. *Computational Mechanics*, submitted.

3 Example

As an example of deep enhanced geothermal reservoirs, we consider a wellbore-doublet system connected to a sub-vertical hydraulic fracture in crystalline rocks (Figure 1). The reservoir is located at depth of 4km where in-situ temperature is around 150°C. The generated hydraulic fracture has a rectangular shape with a length 300m and is inclined with angle of 75° from horizontal. Two wellbores are installed in the system to circulate water mainly through the fracture. Production wellbore connects to the fracture at its middle, assuming the wellbore was used for hydraulic fracturing. Injection wellbore connects to the fracture at 100m below the production.

Properties of in-situ rocks, the hydraulic fracture and fluid are listed in Table 2. Crystalline rock, particularly granite, is assumed as a rock type. The hydraulic fracture is assumed to have aperture of 1mm. Its permeability is determined by Cubic law.

To evaluate hydrothermal processes not only in the fracture but also in surrounding rock matrix, we consider a cubic domain with 900m length, which surrounds the hydraulically most active area, i.e. the hydraulic fracture. Since the domain is symmetric along y axis, only its half space is considered in a simulation model to reduce computation time (Figure 2). We simulate hydrothermal processes in this reservoir for a scenario that fluid with 70°C is injected into the system and circulated with a rate 1 l/s. Initial and boundary conditions applied to the model are summarized in Table 3.

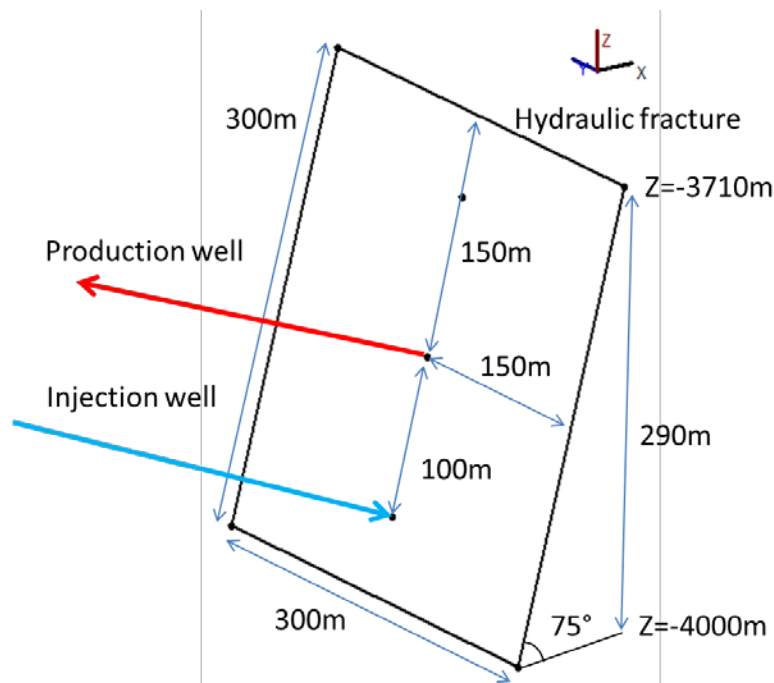


Figure 1 Geometric layout of a fracture and wellbores

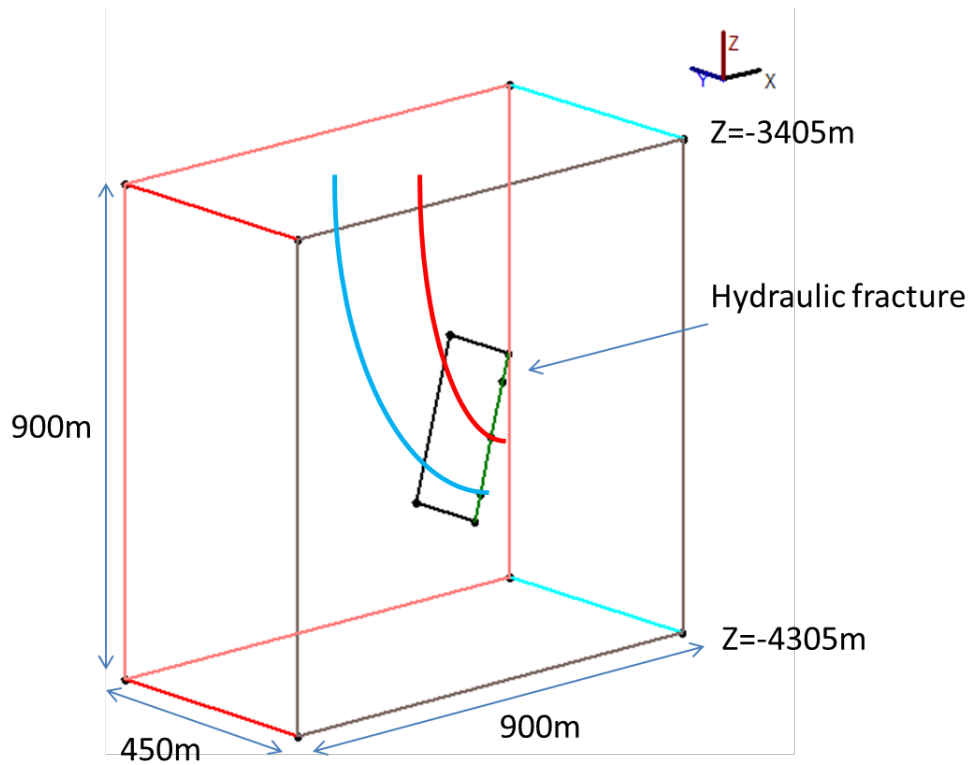


Figure 2 Half symmetric space of geothermal reservoirs (The fracture is located at the center of the domain)

Table 2. Material properties

Rock properties

Porosity	1 %
Permeability	$1.0\text{E-}18 \text{ m}^2$
Specific storage	$1.0\text{E-}10 \text{ Pa}^{-1}$
Density	2600 kg/m^3
Specific heat capacity	850 J/(kg K)
Heat conductivity	3.0 W/(m K)

Fracture properties

Aperture	1 mm
Permeability	$8.33\text{E-}8 \text{ m}^2$
Storage	$1.0\text{E-}10 \text{ Pa}^{-1}$

Fluid properties

Density	1000 kg/m^3
Dynamic viscosity	$3\text{E-}4 \text{ Pa s}$
Specific heat capacity	4000 J/(kg K)
Heat conductivity	0.6 W/(m K)

Table 3 Initial and boundary conditions

Initial conditions

Hydraulic	hydrostatic
-----------	-------------

Thermal	150°C at depth 4km, gradient 30°C/km
<i>Boundary conditions</i>	
Temperature at Injection	70°C
Injection and production rate	1 l/s (1E-3 m ³ /s)
Outer boundary	No flow and heat exchange (i.e. closed system)

4 Pre-processing

In this section, you will learn how to prepare OpenGeoSys input files for the given example using GINA_OGS.

OpenGeoSys requires a set of input files whose names are a combination of the project name and file extensions corresponding to input data type. In this example, we will use “geothermal” as a project name. For instance, input file name for a mesh must be “geothermal.msh”, that for initial conditions is “geothermal.ic”. Table 4 gives a summary of OpenGeoSys input files which are generally required.

Table 4 Required input files

File extension	Description
MSH	Mesh
PCS	Physical process definition
GLI	Geometric object definition used for IC, BC, ST and data output.
IC	Initial conditions
BC	Dirichlet boundary conditions
ST	Source/sink terms (inc. Neumann BC)
MFP	Fluid phase properties
MSP	Solid phase properties
MMP	Medium properties (porous or fracture)
NUM	Numerical settings
TIM	Time stepping control
OUT	Result output settings

4.1 Start-up GINA_OGS

Now you can start GINA_OGS. To easily follow this step-by-step tutorial, please note that there are mainly three areas defined in the user interface (Figure 3):

- Main menu
- Menu for interface to OpenGeoSys files
 - Geometry menu
 - Mesh menu
 - FEM-Parameter menu
 - Calculation/Results menu
- Visualization control

These terms are used in the following tutorial to guide you to access some functionality.

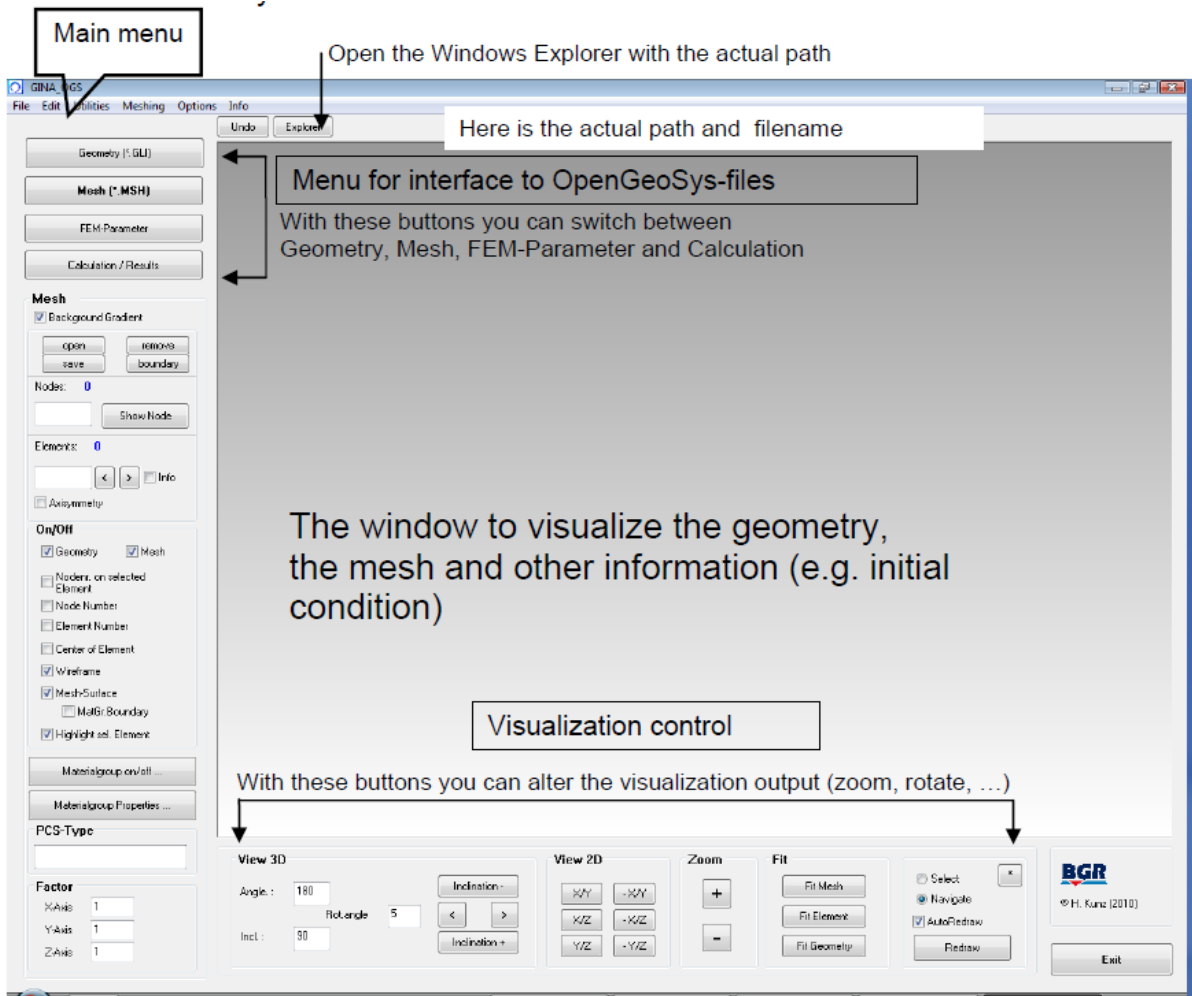


Figure 3 GINA interface

4.2 Meshing (*.msh)

First, we prepare a mesh for a half space of the symmetric reservoir including the hydraulic fracture (Figure 2). Users can generate a mesh using GINA as it provides interface to meshing tools, i.e. Gmsh and Tetgen. It is also possible that users generate a mesh using their own tools if mesh data are provided in the OGS format at the end.

For geothermal reservoir modeling based on the hybrid modeling approach, the following constraints have to be introduced in meshes:

- Fractures are represented by triangle or quadrilateral elements (called fracture elements).
- Rock matrices are represented by volumetric elements, e.g. tetrahedral, hexahedral (called matrix elements).
- Fracture elements must be located along edges of matrix elements.
- Fracture and matrix elements must share element nodes.
- Fracture elements and matrix elements must have different material group id (e.g. 0 for fracture elements and 1 for matrix elements).
- It's recommended to locate nodes at injection and production points to assign boundary conditions.

Figure 4 and Figure 5 show an example mesh using tetrahedral and triangular elements. Elements around the fractures are refined with a node density 5m.

To show a mesh in Gina,

- Go to “Mesh” menu
- Click “Open” on the left panel and select your mesh file.

To navigate a mesh, select “Navigate” option in Visualization control.

- Rotation: drag mouse left button
- Move: drag mouse right button
- Zoom: drag mouse middle button

To show only fracture elements,

- Click “Material group on/of...” on the left panel to open a dialog box.
- Activate only a material group for the fracture (e.g. MG 0) on the dialog box (Figure 6).

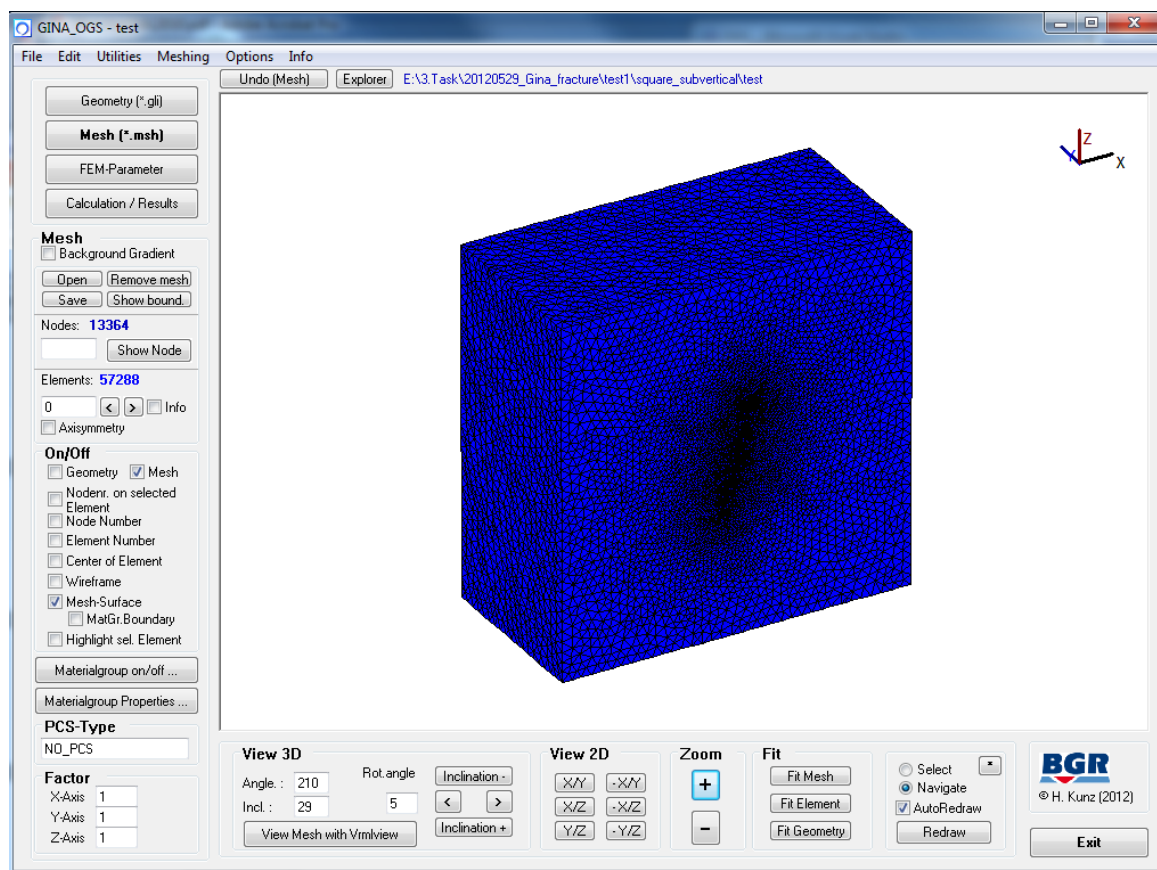


Figure 4 Example mesh: tetrahedrals + triangles

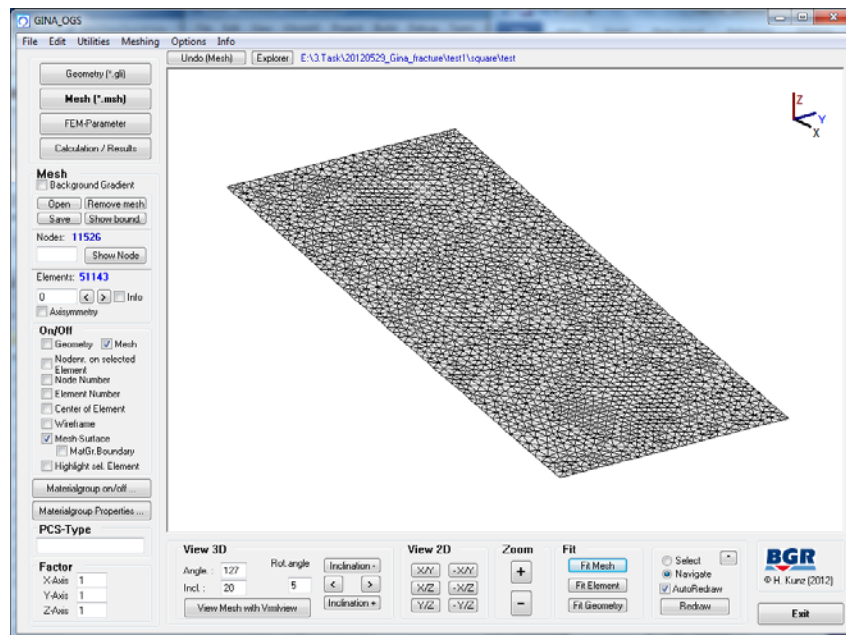


Figure 5 Plot only fracture elements

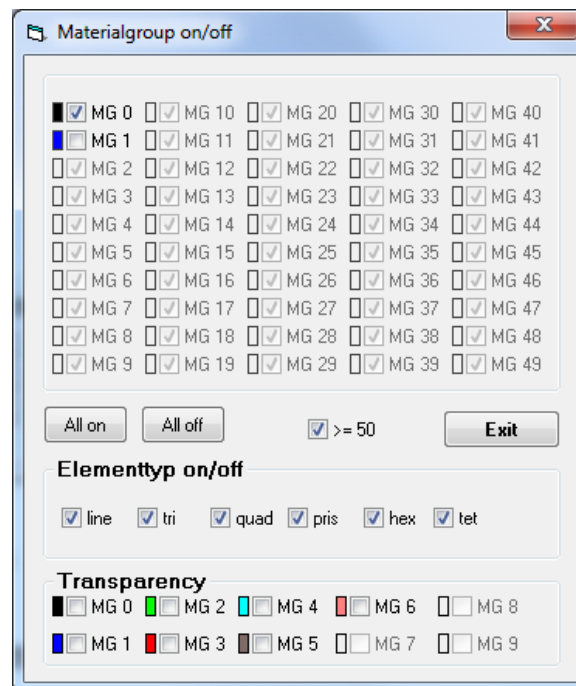


Figure 6 Material group selection

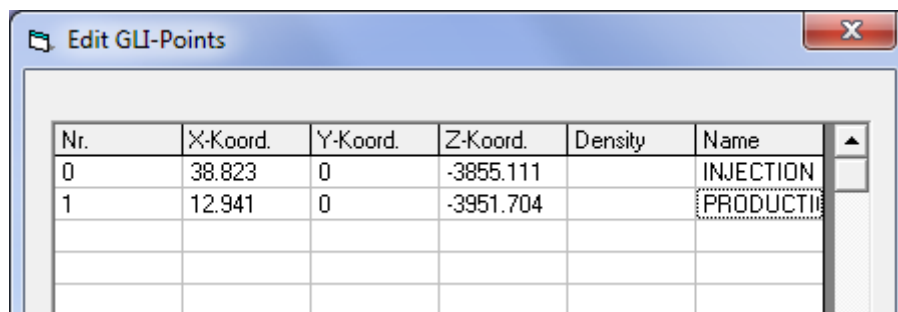
4.3 Geometric object definition (*.gli)

Before creating FEM parameters, we prepare geometric objects which are used later to assign boundary conditions and result output. The following geometric objects are needed in this example,

- Points
 - Injection point
 - Coordinates: (12.941, 0, -3951.704)
 - Name: INJECTION
 - Production point
 - Coordinates (38.823, 0, -3855.111)
 - Name: PRODUCTION

To define these points,

- Go to “Geometry” menu
- Click “Edit Points” on the left panel to open “Edit GLI-Points” dialog.
- Fill two rows in a table as shown in a below figure.
- Click “Apply” and “Exit”.
- Click “Save” with a file name “geothermal.gli”.

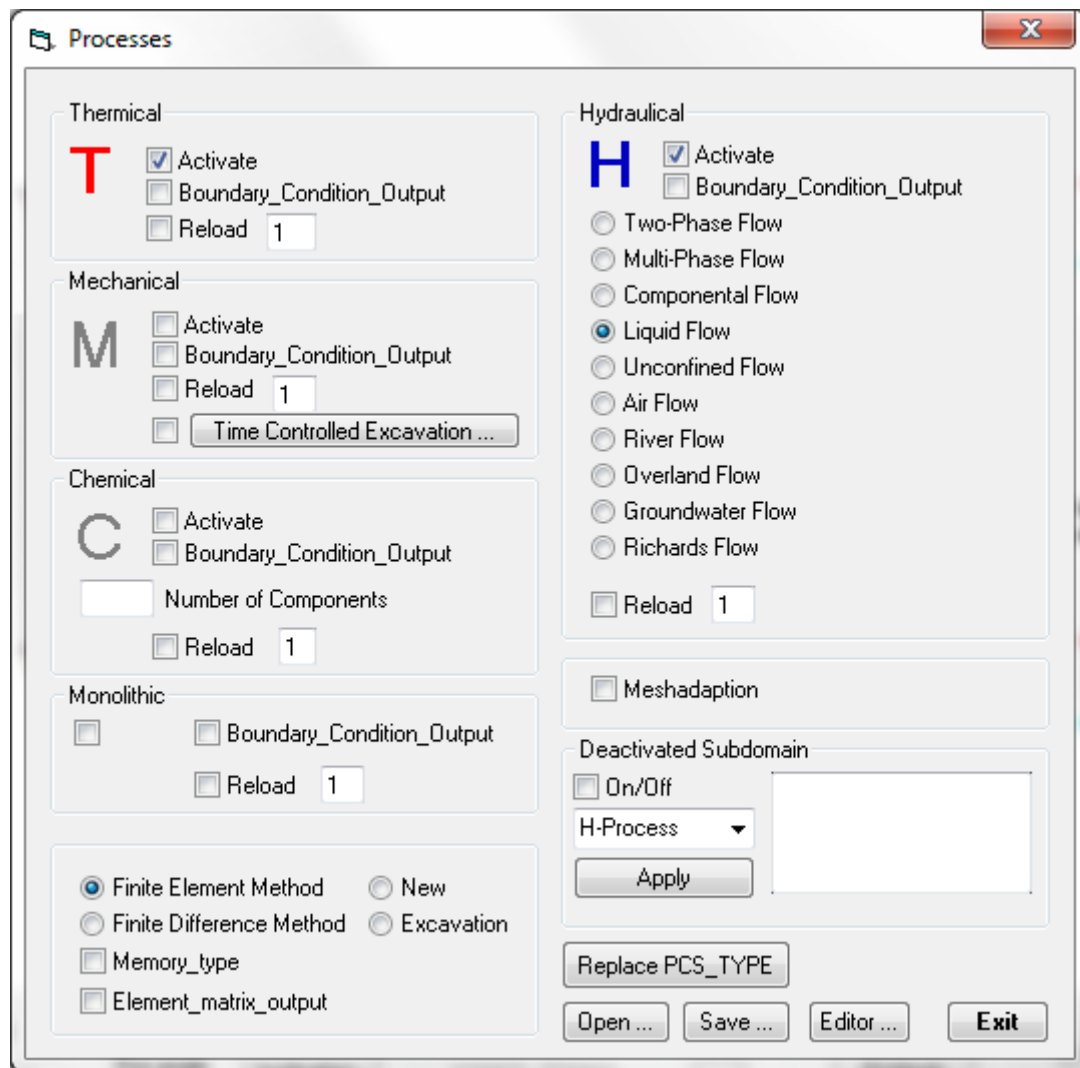


Nr.	X-Koord.	Y-Koord.	Z-Koord.	Density	Name
0	38.823	0	-3855.111		INJECTION
1	12.941	0	-3951.704		PRODUCTION

4.4 Process definition (*.pcs)

Next, we define simulated physical processes.

- Go to “FEM_Parameter” menu, Click “Processes...”
- Activate “Thermal”, “Hydraulic”. Select “Liquid Flow”
- Click “Save...” and “Exit”
- Please check a file “geothermal.pcs” is created in your project directory.



4.5 Initial and boundary conditions

4.5.1 Initial conditions (*.ic)

As described in Section 3, we assume initial hydraulic condition follows hydrostatic. Temperature distribution is linearly dependent on depth with a gradient 30K/km (i.e. 0.3 K/m). Reference is given at depth 4km with temperature 423.5K (150°C).

To define the initial conditions,

- Click "Initial Conditions..."
- Double-click the first row in a table
- Select the following options as shown in Figure to define hydrostatic condition
 - Process: LIQUID_FLOW
 - Variable: PRESSURE
 - Geo_Type: DOMAIN
 - Dis-Type: GRADIENT
 - Value: 0 0 9.81e+3

- Click "Apply" and "Exit"
- Double-click the second row in a table
- Select the following options to define in-situ temperature distribution
 - Process: HEAT_TRANSPORT
 - Variable: TEMPERATURE
 - Geo_Type: DOMAIN
 - Dis-Type: GRADIENT
 - Value: -4000 423.15 0.03
- Click "Apply" and "Exit"
- Click "Exit" and save.

Remark

- Expression of value for GRADIENT is (reference z coordinates) (reference value) (gradient).

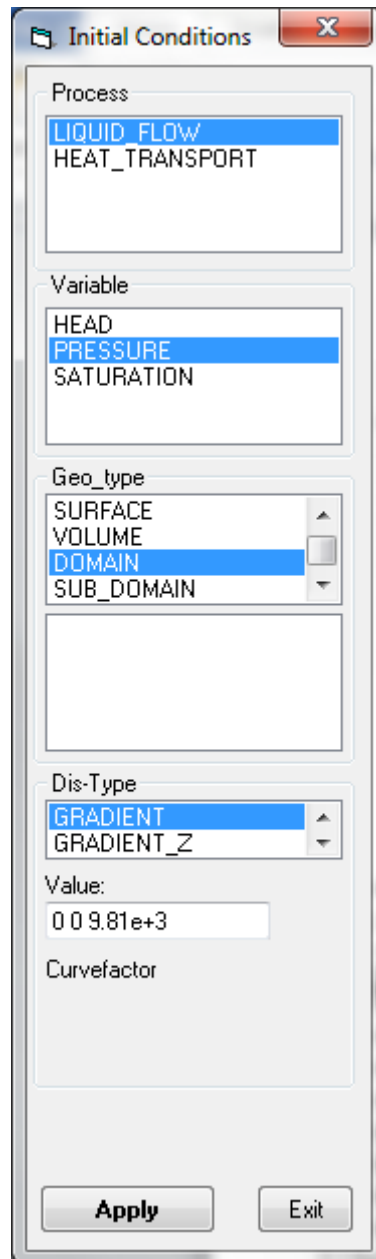


Figure 7 Initial condition for hydrostatic

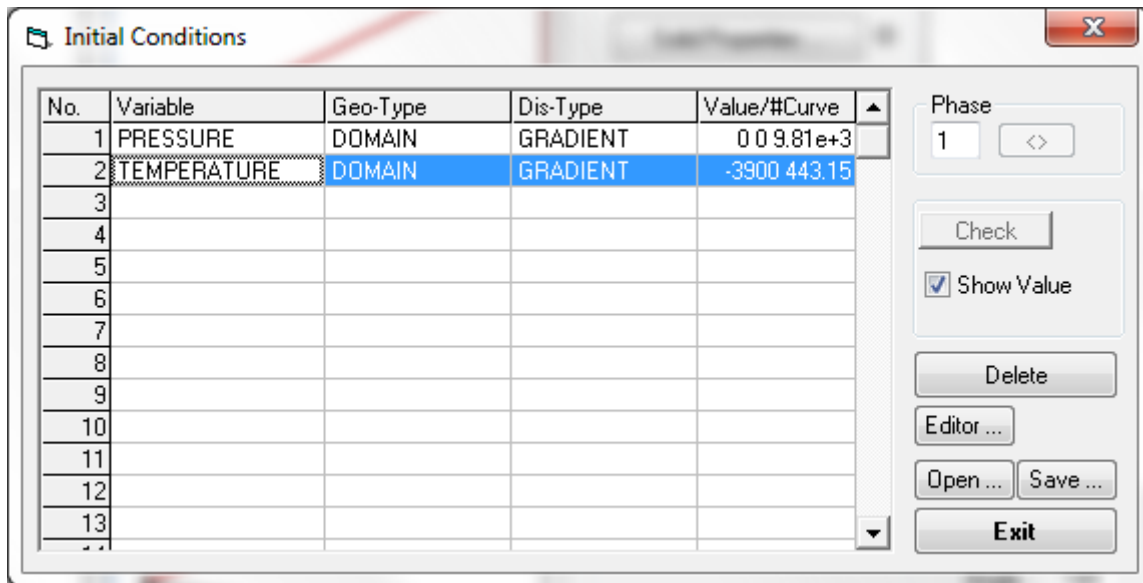


Figure 8 List of initial conditions defined

4.5.2 Dirichlet boundary conditions (*.bc)

Next, we define Dirichlet boundary conditions. In this example, we have fixed temperature 343.15K (70°C) at the injection point.

To define the B.C.,

- Click “Boundary Conditions...”
- Double-click the first row in a table
- Select the following options to define injected fluid temperature
 - Process: HEAT_TRANSPORT
 - Variable: TEMPERATURE
 - Geo_Type: POINT, INJECTION
 - Dis-Type: CONSTANT
 - Value: 343.15
- Click “Apply” and “Exit”
- Click “Exit” and save

Remarks

- Any geometric objects used for Geo_Type have to be defined in GLI files.

4.5.3 Source Terms inc. Neumann BC (*.st)

Neumann boundary conditions and any source and sinks are defined in Source Terms files (*.ST).

To define injection and production rate 1 l/s for a half symmetric space, i.e. 0.5×10^{-3} m³/s ,

- Click “Source Terms...”
- Double-click the first row in a table
- Select the following options to define injection rate
 - Process: LIQUID_FLOW

- Variable: PRESSURE
 - Geo_Type: POINT, INJECTION
 - Dis-Type: CONSTANT
 - Value: 0.5e-3
- Click “Apply” and “Exit”
- Double-click the second row in a table
- Select the following options to define production rate (negative sign!)
 - Process: LIQUID_FLOW
 - Variable: PRESSURE
 - Geo_Type: POINT, PRODUCTION
 - Dis-Type: CONSTANT
 - Value: -0.5e-3
- Click “Apply” and “Exit”
- Click “Exit” and save

4.6 Material properties

In the following, we define physical properties of geothermal fluid, rock matrix and the hydraulic fracture which are described in Section 3. OpenGeoSys assumes that entire space is filled by porous media. Users should separately prepare input files for setting properties of fluid phase, solid phase and medium, respectively.

OpenGeoSys allows users to have different types of materials (e.g. rock type) using material id (e.g. 0, 1) as an attribute of each mesh element. Users can set individual solid phase and medium properties for each material type.

4.6.1 Fluid phase properties (*.mfp)

First, we define properties of geothermal fluid (brine). It is assumed fluid density and dynamic viscosity varies with pressure, temperature and salinity. Salinity is constant 60g/l, in this example. Specific heat and thermal conductivity is constant during simulations.

To create an input file for fluid properties,

- Click “Fluid Properties...” to open Edit dialog box.
- Copy and paste the following in a text box in the dialog.
- Click “Exit” and save.

```
GeoSys-MFP: Material Fluid Properties -----
#FLUID_PROPERTIES
$FLUID_TYPE
  LIQUID
$DAT_TYPE
  LIQUID
$DENSITY
  1 1000
$VISCOSITY
  1 3e-4
$SPECIFIC_HEAT_CAPACITY
  1 4000
$HEAT_CONDUCTIVITY
  1 0.6
#STOP
```

Remark

- For details of available options, please refer to OpenGeoSys manual.

4.6.2 Solid phase properties (*.msp)

Next, we set solid phase properties for all materials. In this example, we have two types of materials, i.e. the hydraulic fracture and surrounding rock.

To define solid properties,

- Click “Solid Properties...” to open Edit dialog box.
- Copy and paste the following in a text box in the dialog.
- Click “Exit” and save.

```

GeoSys-MSP: Material Solid Properties -----
; fracture
#SOLID_PROPERTIES
$DENSITY
  1 2600
$THERMAL
  EXPANSION
    6.0e-6
  CAPACITY
    1 850
  CONDUCTIVITY
    1 3.0
; matrix
#SOLID_PROPERTIES
$DENSITY
  1 2600
$THERMAL
  EXPANSION
    6.0e-6
  CAPACITY
    1 850
  CONDUCTIVITY
    1 3.0
#STOP

```

Remark

- Users should define solid properties for each material type in an order of material id.
- The definition for each material begins with keyword #SOLID_PROPERTIES.

4.6.3 Medium properties (*.mmp)

We set medium properties such as porosity and permeability for all materials as we did for solid properties.

To define medium properties,

- Click "Medium Properties..."
- Click "Editor ..." to open Edit dialog box.
- Copy and paste the following in a text box in the dialog.
- Click "Exit" and save.

```

GeoSys-MMP:
;fracture
#MEDIUM_PROPERTIES
$GEOMETRY_DIMENSION
  2
$GEOMETRY_AREA
  1e-3
$POROSITY
  1 1.0
$STORAGE
  1 1e-10
$TORTUOSITY
  1 1.0
$PERMEABILITY_TENSOR
  ISOTROPIC 8.33e-8
$MASS_DISPERSION
  1 1.0 1.0
;matrix
#MEDIUM_PROPERTIES
$GEOMETRY_DIMENSION
  3
$GEOMETRY_AREA
  1.0
$POROSITY
  1 0.01
$STORAGE
  1 1e-10
$TORTUOSITY
  1 1.0
$PERMEABILITY_TENSOR
  ISOTROPIC 1.e-18
$MASS_DISPERSION
  1 1.0 1.0
#STOP

```

Remark

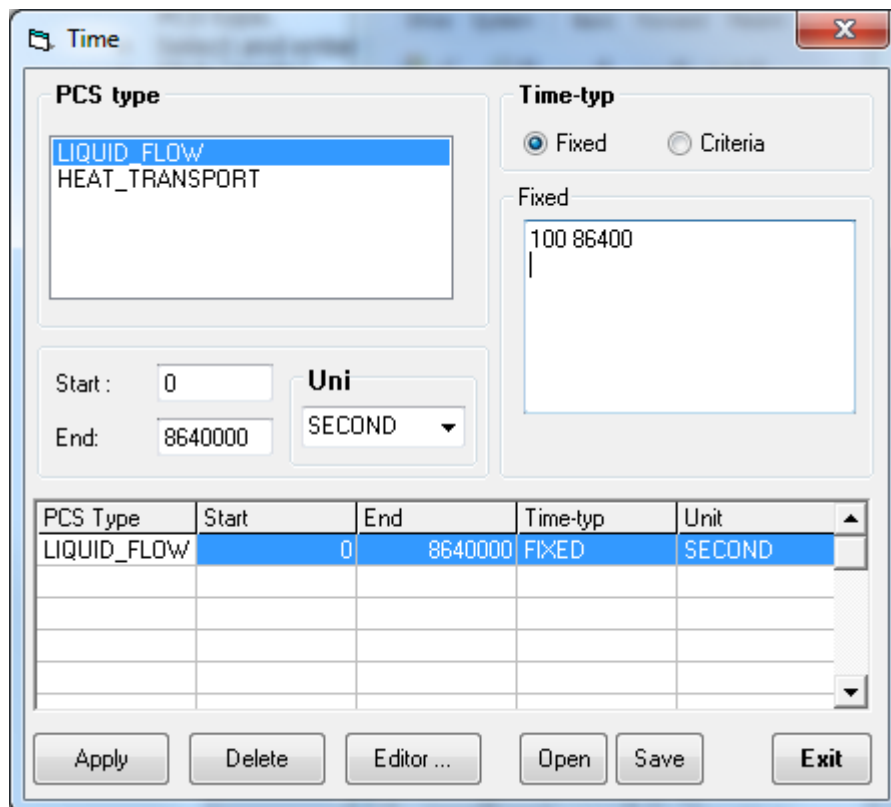
- The definition for each material begins with a keyword `#MEDIUM_PROPERTIES`.
- `$GEOMETRY_DIMENSION` is used to define dimension of setting medium properties such as permeability. In this example, for the fracture, it is set to 2 so that one can define permeability tensor in local coordinates of 2D fracture elements. OpenGeoSys internally transform it to 3D.
- Users set fracture aperture to `$GEOMETRY_AREA`. In this example, the aperture is 1 mm.

4.7 Time stepping (*.tim)

Time stepping control can be configured in TIM files. In this example, we use a time step size 1 day for total simulation period 100 days.

To define time stepping,

- Click "Time..."
- Select LIQUID_FLOW from a list in PCS type.
- Select and enter the following options
 - Start: 0
 - End: 8640000
 - Unit: SECONDS
 - Time-type: Fixed, 100 86400
- Click "Apply"
- Click the second row in the bottom table, and select HEAT_TRANSPORT as PCS type.
- Select and enter the same options as LIQUID_FLOW.
- Click "Apply"
- Click "Exit" and save.



4.8 Other settings

4.8.1 Numerical properties (*.num)

Next, we configure numerical options for time discretization, linear solver, nonlinear method, and coupling method. In this example, we use Backward Euler method for time discretization, BiCGstab with Jacobi preconditioner for linear solver. Coupling of hydraulic and thermal processes is achieved in a partitioned way.

To define numerical properties,

- Click “Numerical Properties...” to open Edit dialog box.
- Copy and paste the following in a text box in the dialog.
- Click “Exit” and save.

```

$OVERALL_COUPLING
;min_iter -- max_iter
1      1
#NUMERICS
$PCS_TYPE
  LIQUID_FLOW
$LINEAR_SOLVER
; method error_tolerance max_iterations theta precondition storage
2      6 1.e-011      3000      1.0 1      2
$NON_LINEAR_ITERATIONS
;type -- error_method -- max_iterations -- relaxation -- tolerance(s)
PICARD LMAX 1 0.0 1e-3
$COUPLING_CONTROL
;error method -- tolerances
  LMAX 1.e-3
#NUMERICS
$PCS_TYPE
  HEAT_TRANSPORT
$LINEAR_SOLVER
; method error_tolerance max_iterations theta precondition storage
2      6 1.e-011      3000      1.0 1      4
$NON_LINEAR_ITERATIONS
PICARD LMAX 1 0.0 1e-3
;type -- error_method -- max_iterations -- relaxation -- tolerance(s)
$COUPLING_CONTROL
;error method -- tolerances
  LMAX 1.e-3
#STOP

```

4.8.2 Output Parameters (*.out)

To output simulation results to files,

- Click “Output Parameters...”.
- Double-click the first row in a table to show Output dialog.
- To output each time step results of entire domain in PVD/VTU file formats, select and enter the following options as shown in Figure.
 - PCS type: All processes
 - Node values: PRESSURE1, VELOCITY_X1, VELOCITY_Y1, VELOCITY_Z1, TEMPERATURE1
 - Geo-type: DOMAIN
 - Tim-type: Steps, 1
 - Dattype: PVD
- Click “Apply” and exit.
- To output simulated pressure and temperature at production point in

TECPLOT file format, select and enter the following options as shown in Figure.

- PCS type: All processes
- Node values: PRESSURE1, TEMPERATURE1
- Geo-type: POINT, PRODUCTION
- Tim-type: Steps, 1
- Dattype: Tecplot
- Click "Apply" and exit.
- Click "Exit" and save.

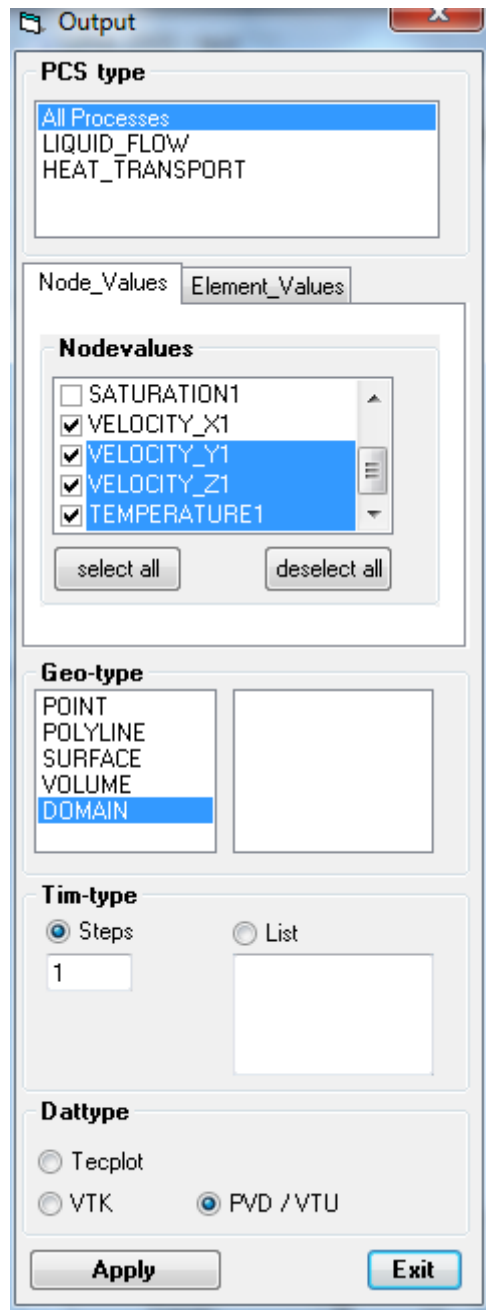


Figure 9 Output results in entire domain

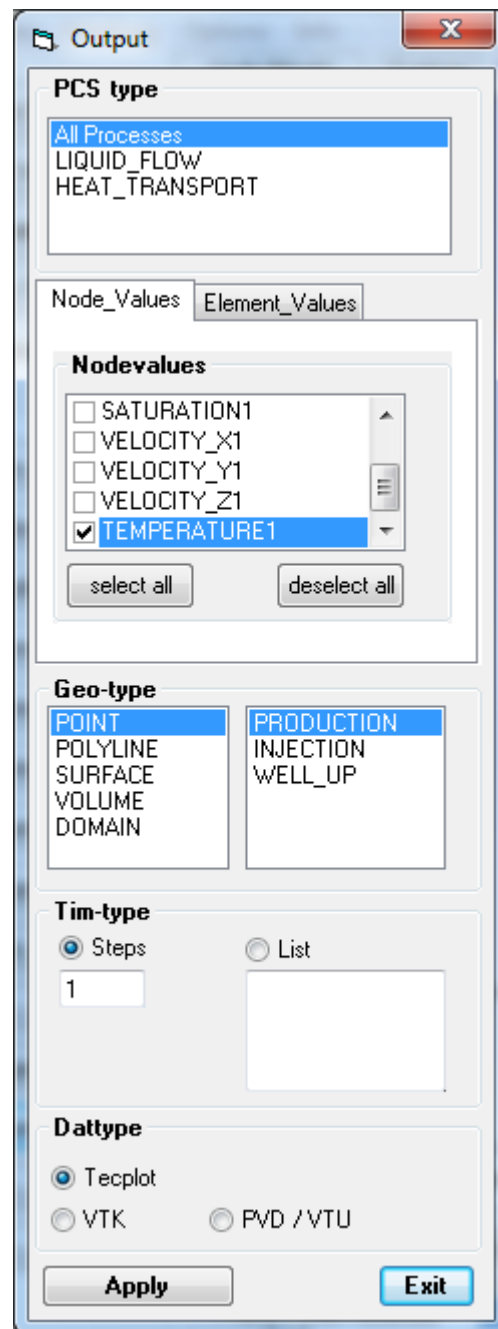


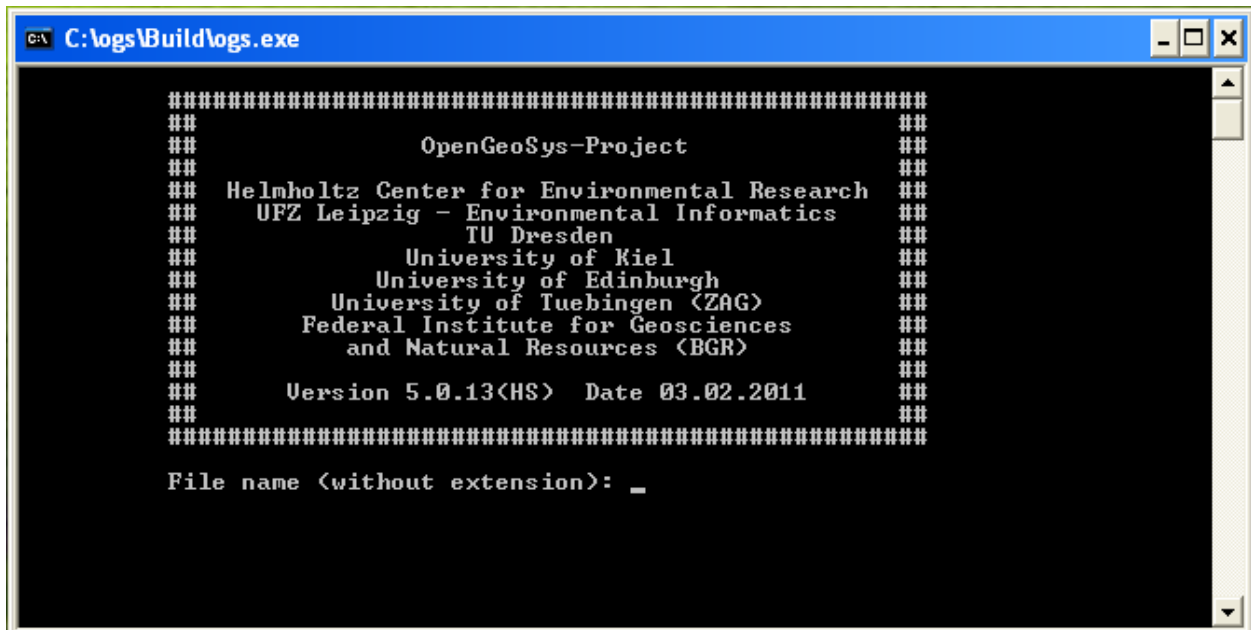
Figure 10 Output results at the production point

5 Simulation

Before starting simulations, please check confirm that you have prepared the following input files in the same directory.

- geothermal.bc
- geothermal.gli
- geothermal.ic
- geothermal.mfp
- geothermal.mmp
- geothermal.msh
- geothermal.msp
- geothermal.num
- geothermal.out
- geothermal.pcs
- geothermal.st
- geothermal.tim

To start simulations, click the executable ogs.exe.



At the top of this Pop-up, location of the executable file is given. 5.0.13(HS) tells that used executable is from the version number 5.0.13 and was taken by HS on date 03.02.2011. This asks initial name of the input files with its location.

Assuming you have the above input files in C:\logs, type C:\logs\geothermal

Press the ENTER

Then simulation starts and continues until the time you specified in TIM file. At the end of this, simulation output file(s) asked in the OUT file will appear at the location of input files, i.e. C:\logs\geothermal.

6 Post-processing with ParaView

We visualize simulation results using ParaView.

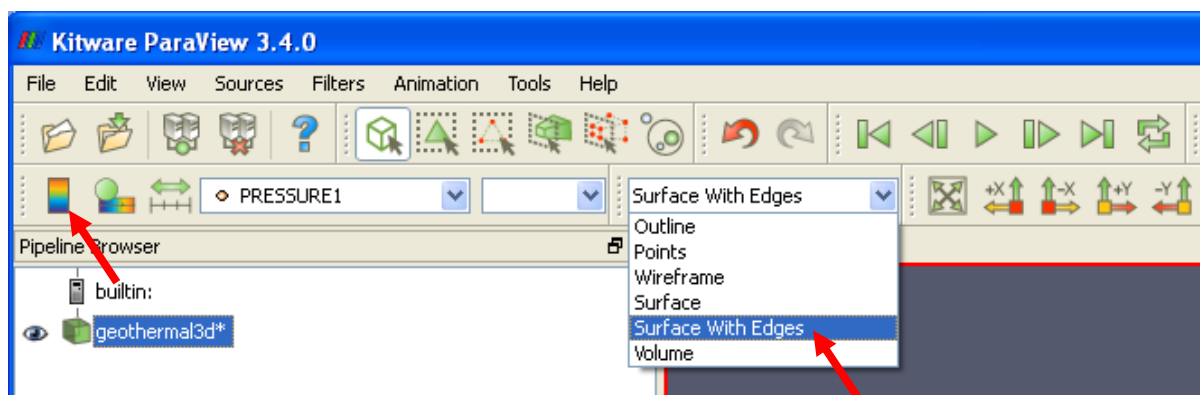
6.1 Load PVD/VTU files

Start ParaView from Start menu and load result files (geothermal.pvd). Click the File Open button or click "File" menu > "open".

Browse the directory where result files exist. Select "geothermal.pvd".

Click "Apply" button on the Object Inspector view to show results.

You can display mesh edges and legends.



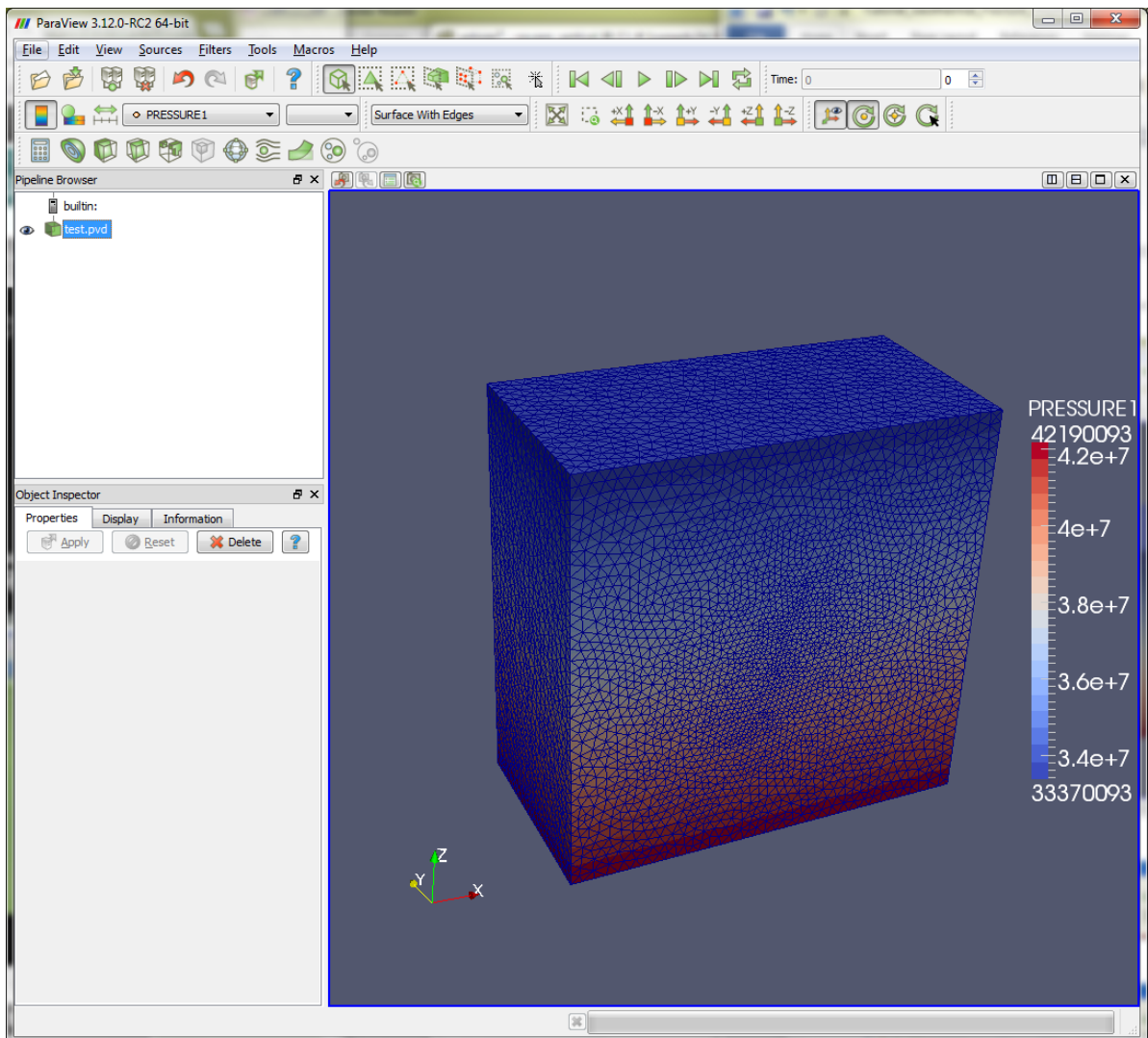
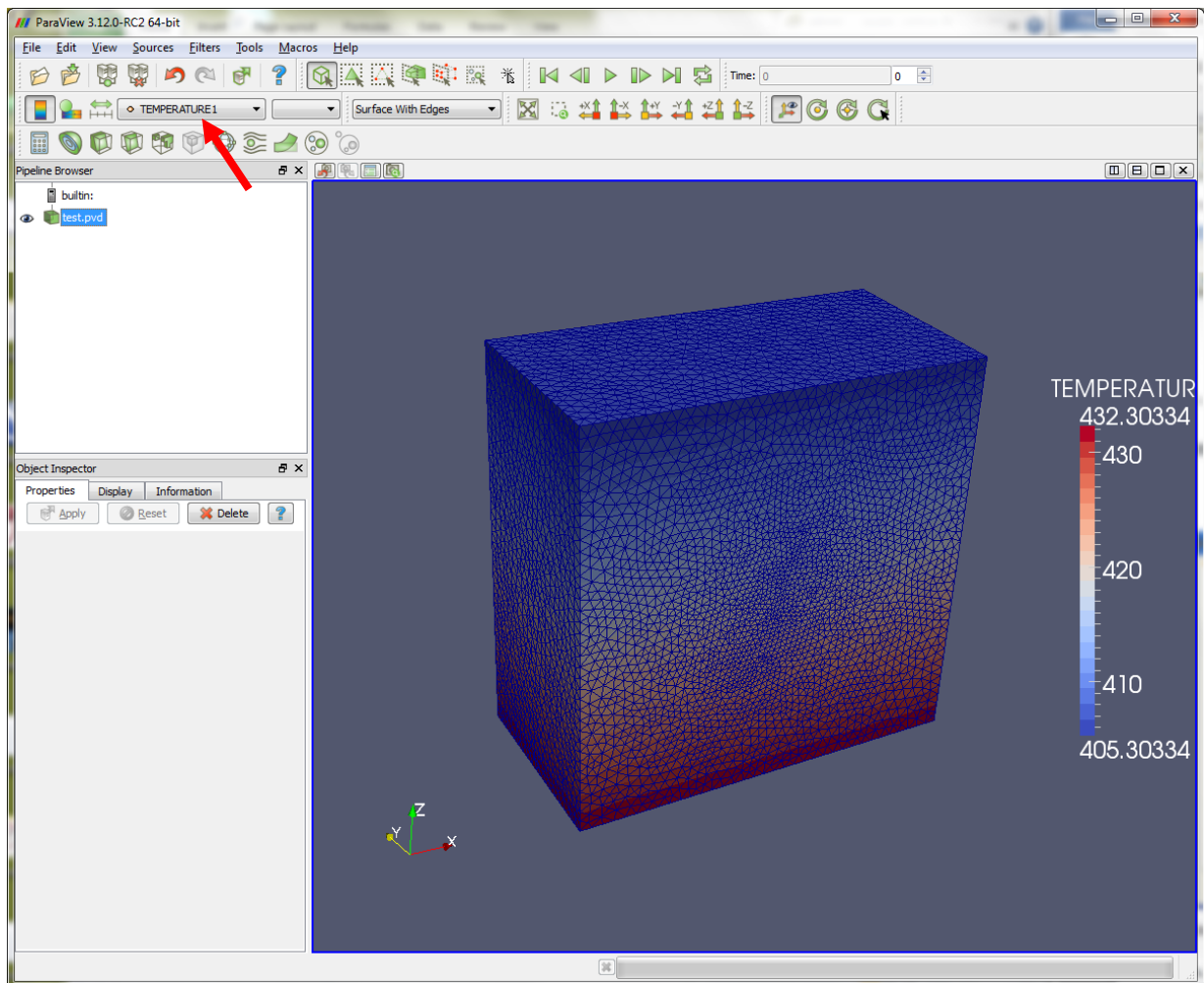
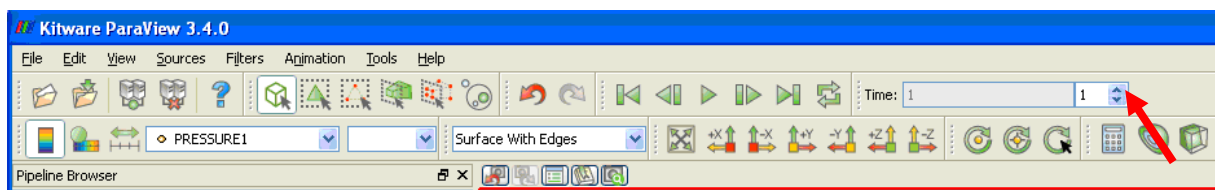


Figure 11 3D plot with legend

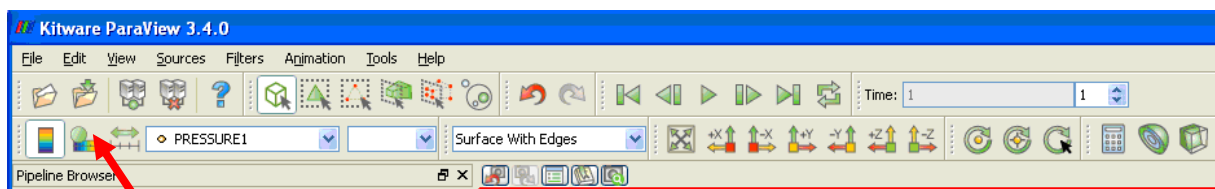
To plot temperature distribution, change visualizing data set on the tool bar from "PRESSURE1" to "TEMPERATURE1".

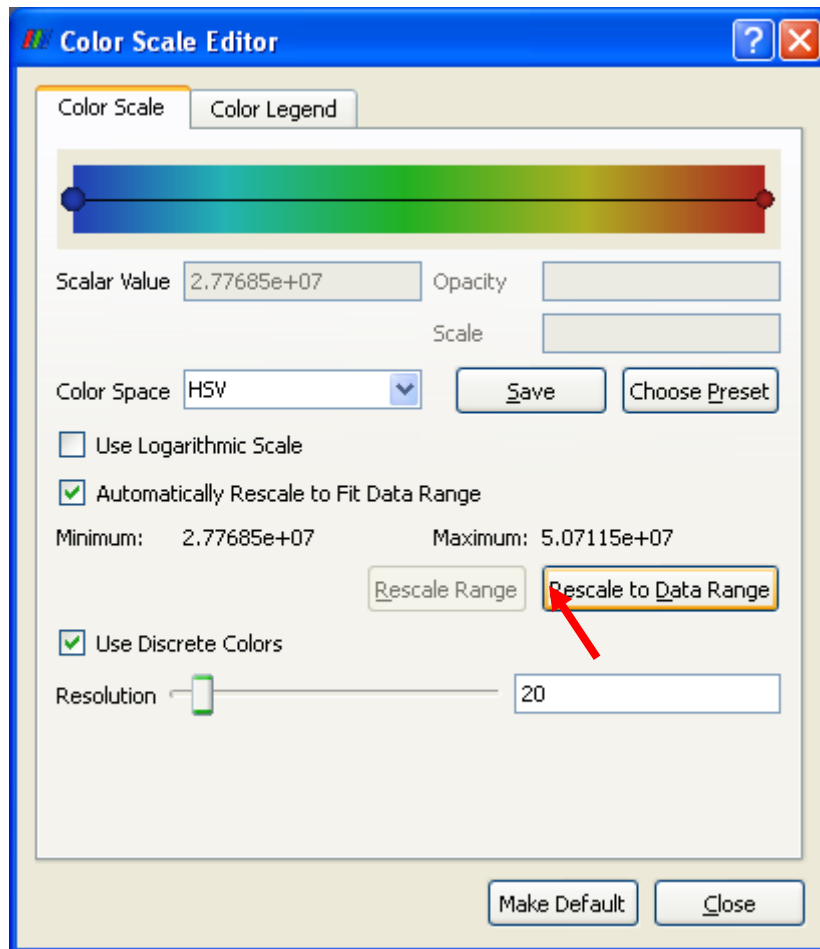


Show results of 1st time step.



Change color scale.

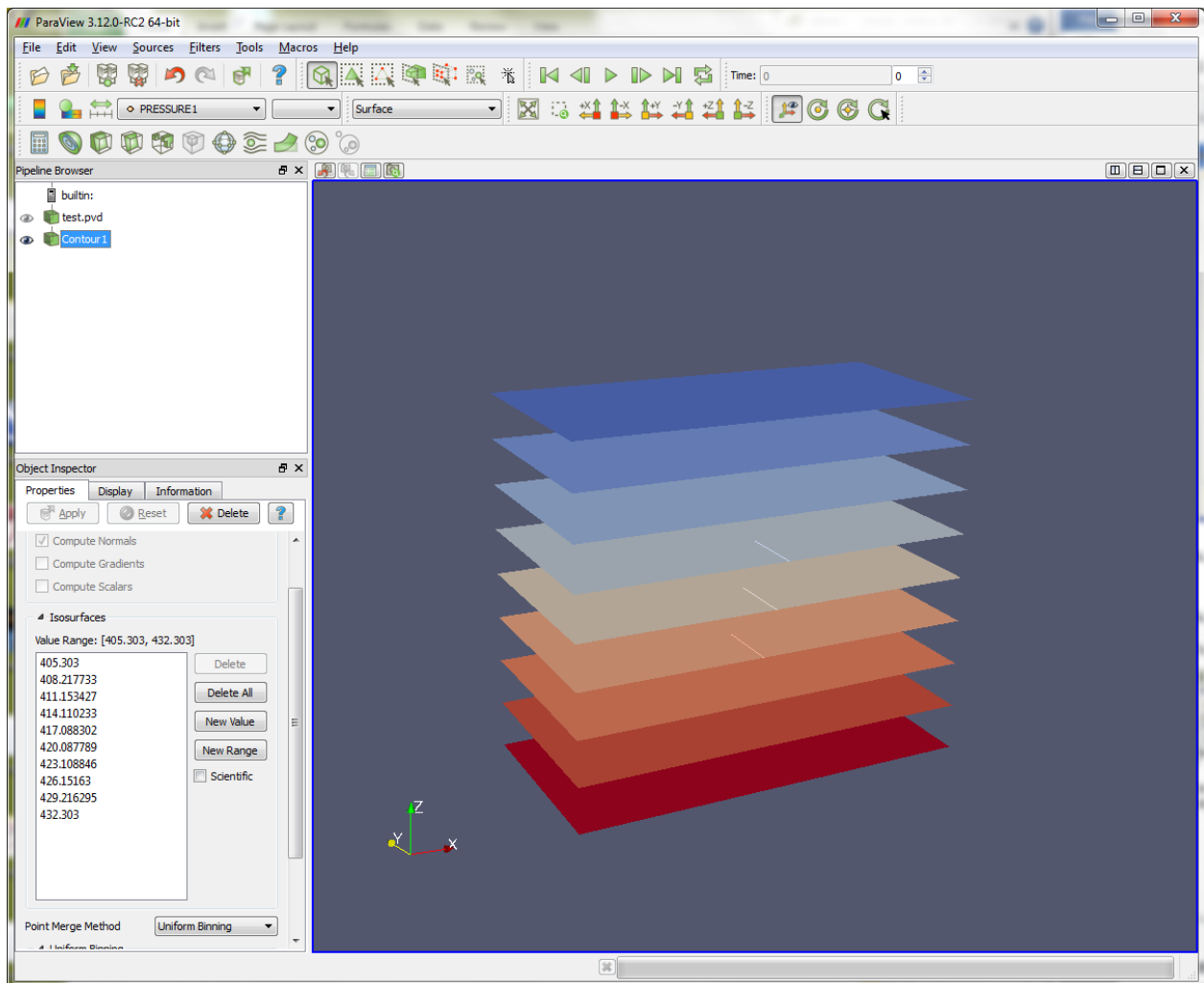




6.2 Plot temperature isosurface

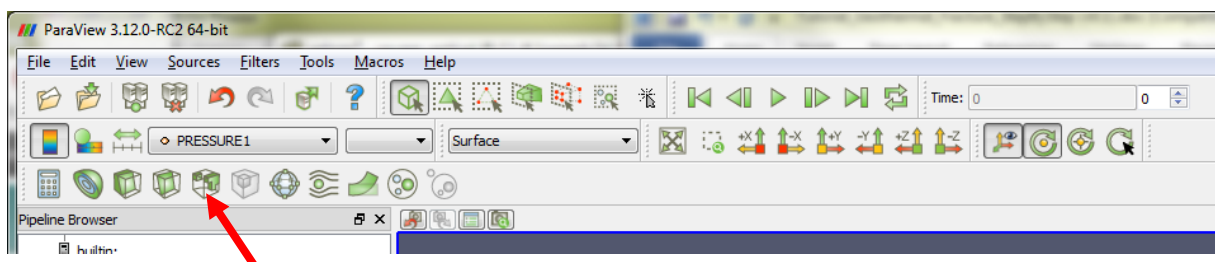
Follow below procedures to show temperature isosurface,

- Select geothermal.pvd on Pipeline Browser.
- Click Contour filter.
- Select Contour by TEMPERATURE1. Click "Delete All" and "New Range". Click "OK".
- Click "Apply"

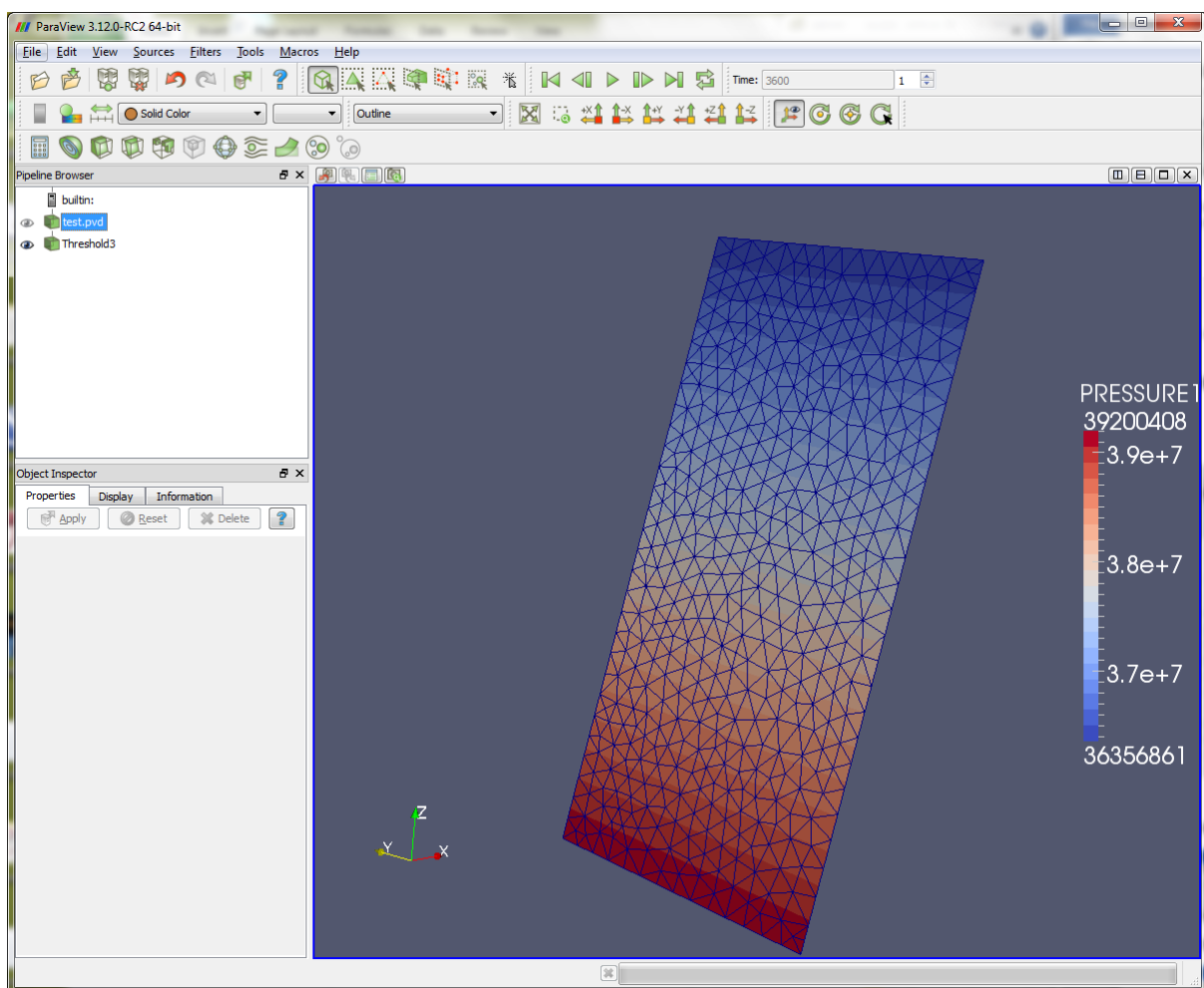
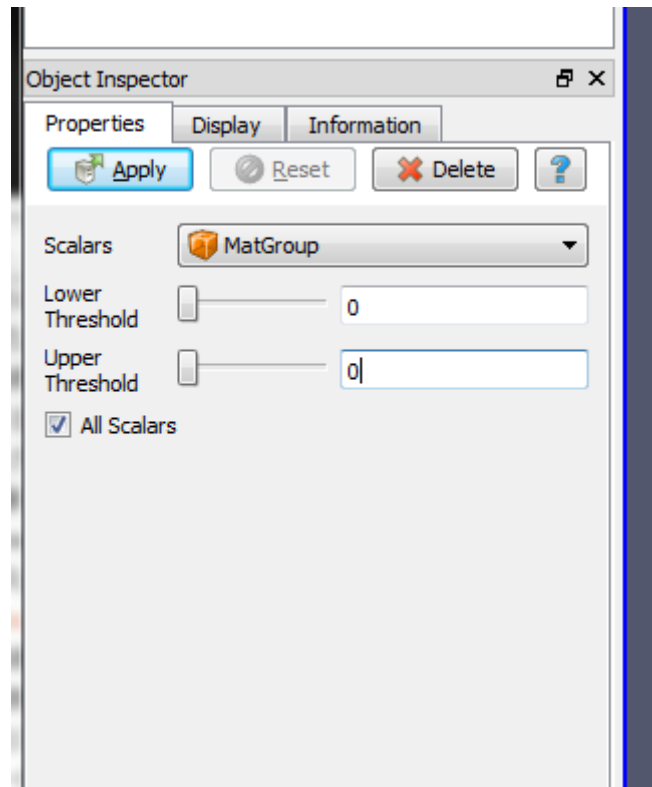


6.3 Plot only fracture elements

To show only the fracture elements, click Threshold filter on toolbar.

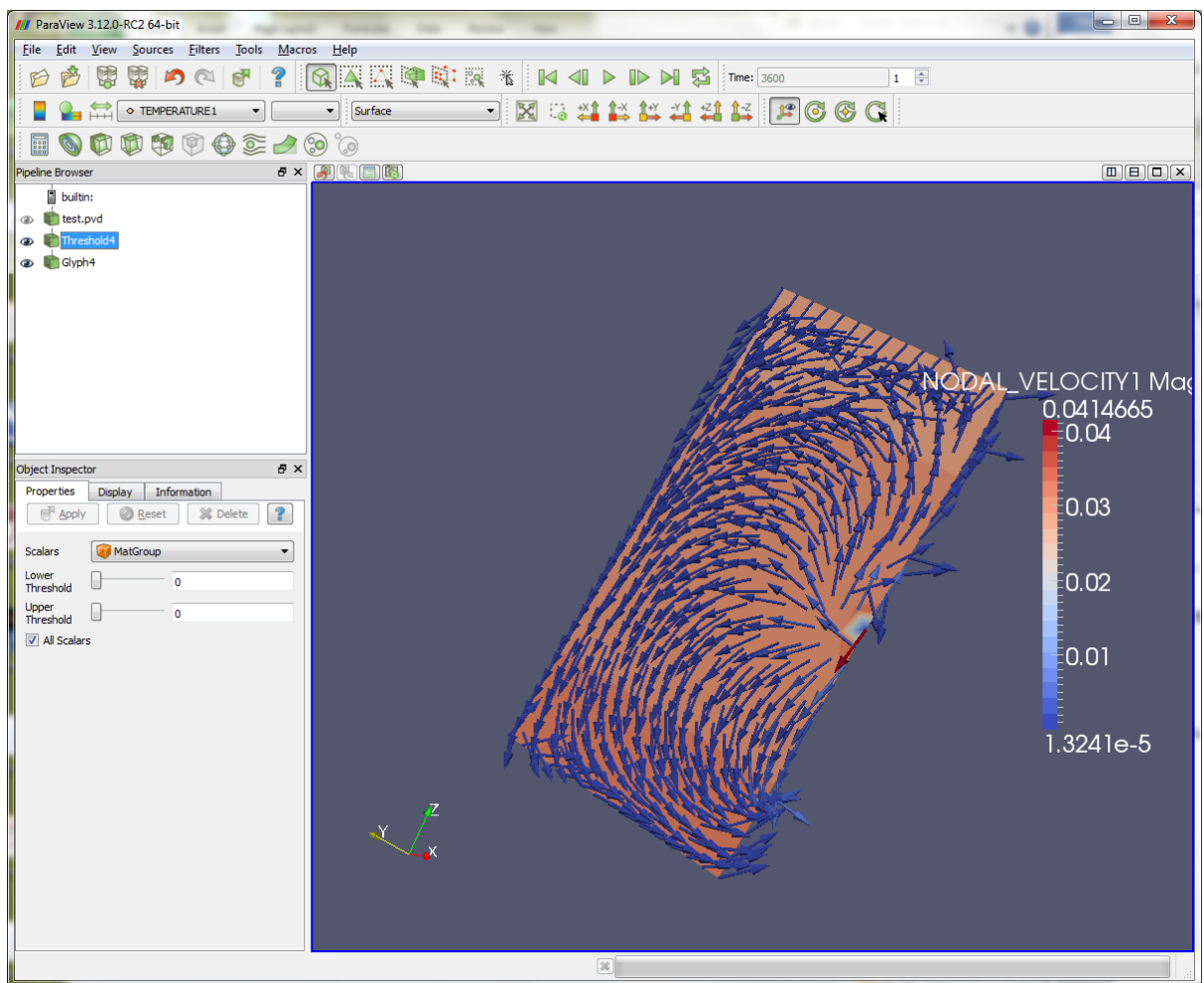
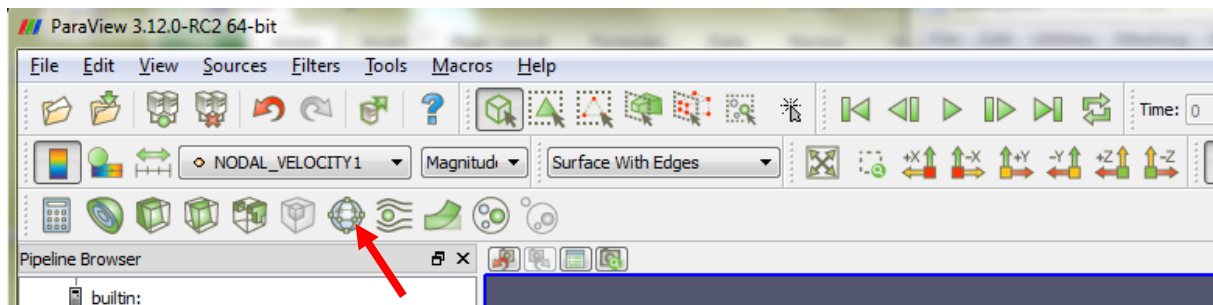


On Properties tab for the filter, select MatGrop in Scalars and set Upper Threshold to 0. Click "Apply".



6.4 Plot flow vector in fracture elements

- Select the generated Threshold pipeline to plot fracture elements in Pipeline Browser.
- Click Glyph filter on toolbar.
- In Object Inspector, select NODAL_VELOCITY1 to Vectors, set Scale Model "off". Click "Apply"



7 Advanced topics

7.1 Variable fluid properties

Geothermal fluid is often brine. Its properties, particularly density and viscosity, actually vary corresponding to changes in temperature, pressure as shown in Figure 12 and Figure 13.

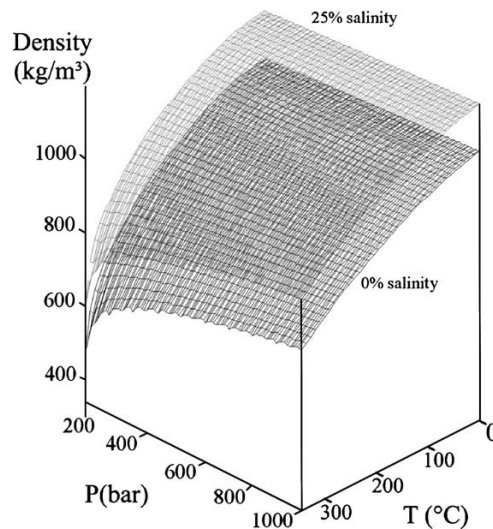


Figure 12 Density of water (McDermott et al. 2006, after IAPWS-IF97)

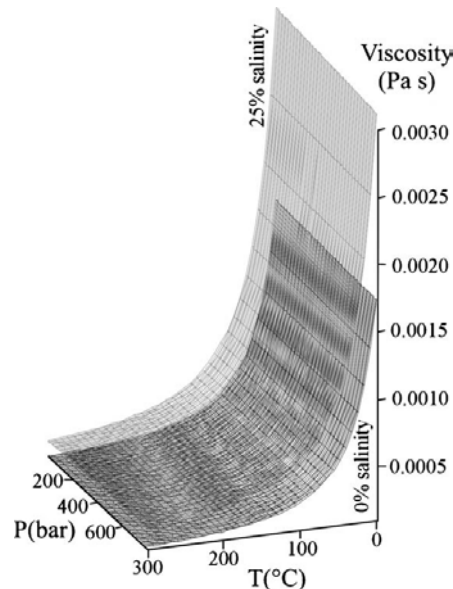


Figure 13 Viscosity of water (McDermott et al. 2006, after Hewlett-Packard Petroleum Fluids pac.)

There are several ways to model equation of state (EoS) for the fluid. For example, McDermott et al. (2006) presented nonlinear EoS depending on temperature, pressure and salinity mainly based on IAPWS-IF97. An example of a MFP file to use this EoS is shown below for a situation in Table 5.

Table 5 Nonlinear fluid properties*Fluid properties*

Salinity	60 g/l
Density	$f(p, T, \text{salinity})$: McDermott et al. 2006
Dynamic viscosity	$f(p, T, \text{salinity})$: McDermott et al. 2006
Specific heat capacity	4000 J/(kg K)
Heat conductivity	0.6 W/(m K)

```

GeoSys-MFP: Material Fluid Properties -----
#FLUID_PROPERTIES
$FLUID_TYPE
  LIQUID
$DAT_TYPE
  LIQUID
$DENSITY
  8 60.0
$VISCOSITY
  7 60.0
$SPECIFIC_HEAT_CAPACITY
  1 4000
$HEAT_CONDUCTIVITY
  1 0.6
#STOP

```

Please note that using variable fluid properties requires use of nonlinear solvers. Replace your NUM file with the following.

```

$OVERALL_COUPLING
;min_iter -- max_iter
1      25
#NUMERICS
$PCS_TYPE
LIQUID_FLOW
$LINEAR_SOLVER
; method error_tolerance max_iterations theta precondition storage
2      6 1.e-011      3000      1.0 1      2
$NON_LINEAR_ITERATIONS
;type -- error_method -- max_iterations -- relaxation -- tolerance(s)
PICARD LMAX 100 0.0 1e-3
$COUPLING_CONTROL
;error method -- tolerances
LMAX 1.e-3
#NUMERICS
$PCS_TYPE
HEAT_TRANSPORT
$LINEAR_SOLVER
; method error_tolerance max_iterations theta precondition storage
2      6 1.e-011      3000      1.0 1      4
$NON_LINEAR_ITERATIONS
PICARD LMAX 100 0.0 1e-3
;type -- error_method -- max_iterations -- relaxation -- tolerance(s)
$COUPLING_CONTROL
;error method -- tolerances
LMAX 1.e-3
#STOP

```

7.2 Reducing numerical oscillations in heat transport

We often face numerical oscillations when modeling advection-dominated heat transport processes. In principle, one can reduce them by adjusting time step size and spatial discretization in a mesh so that it satisfies some stability criteria such as Courant number. However, in practice, it is difficult to find optimal settings to satisfy the criteria due to inhomogeneous fluid velocity and nonlinear fluid properties. Therefore it is common practice to apply stabilization techniques such as Unwinding method.

OpenGeoSys provides two stabilization techniques: mass lumping method and Streamline-Upwind/Petrov-Galerkin (SUPG) method. Mass lumping method can be activated with a keyword \$ELE_MASS_LUMPING. SUPG method can be activated with a keyword \$ELE_SUPG. It is possible to combine both methods. An example NUM file to use the both methods is given below.

```

$OVERALL_COUPLING
;min_iter -- max_iter
1      1
#NUMERICS
$PCS_TYPE
  LIQUID_FLOW
$LINEAR_SOLVER
; method error_tolerance max_iterations theta precondition storage
2      6 1.e-011      3000      1.0 1      2
$NON_LINEAR_ITERATIONS
;type -- error_method -- max_iterations -- relaxation -- tolerance(s)
  PICARD LMAX 1 0.0 1e-3
$COUPLING_CONTROL
;error method -- tolerances
  LMAX 1.e-3
#NUMERICS
$PCS_TYPE
  HEAT_TRANSPORT
$LINEAR_SOLVER
; method error_tolerance max_iterations theta precondition storage
2      6 1.e-011      3000      1.0 1      4
$NON_LINEAR_ITERATIONS
  PICARD LMAX 1 0.0 1e-3
;type -- error_method -- max_iterations -- relaxation -- tolerance(s)
$COUPLING_CONTROL
;error method -- tolerances
  LMAX 1.e-3
$ELE_MASS_LUMPING
1
$ELE_SUPG
1 0 0
;method length_mode(0: max,1: min,2: avg) diffusivity_mode(0: max,1: min,2: mag)
#STOP

```

7.3 Output elemental velocity

In the above tutorial, OpenGeoSys outputs Darcy velocity at nodes into files. Nodal velocity is calculated by taking an average of integration point values of adjacent elements. If users would like to know elemental velocity which is more accurate than nodal velocity, please change OUT files as it includes \$ELE_VALUES and VELOCITY1_X as shown below.

```
#OUTPUT
$NOD_VALUES
  PRESSURE1
  VELOCITY_X1
  VELOCITY_Y1
  VELOCITY_Z1
  TEMPERATURE1
$ELE_VALUES
  VELOCITY1_X
$GEO_TYPE
  DOMAIN
$DAT_TYPE
  PVD
$TIM_TYPE
  STEPS 1
#STOP
```

7.4 Steady-state modeling

OpenGeoSys can simulate steady-state for hydraulic and transport processes. To do it, users should set \$TIM_TYPE to STEADY in each steady-state process definition in PCS files. Example for liquid flow is given below.

```
#PROCESS
$PCS_TYPE
  LIQUID_FLOW
$TIM_TYPE
  STEADY
#STOP
```

* STEADY is not supported in LIQUID_FLOW! Instead, please set all storage coefficients to zero in MMP files.

7.5 Conservative tracer transport modeling

It is possible to simulate tracer transport in addition to hydrothermal processes. A governing equation for conservative solute transport process in porous medium can be expressed as

$$n \frac{\partial C}{\partial t} + \nabla \cdot (\mathbf{q}C - \mathbf{D} \nabla C) = nQ_c$$

where C is a dissolved concentration, \mathbf{D} is a diffusion-dispersion tensor, and Q_c is source/sink. The diffusion-dispersion tensor is defined as

$$\mathbf{D} = (n\tau D_m + \alpha_L \|\mathbf{q}\|) \mathbf{I} + (\alpha_L - \alpha_T) \frac{\mathbf{q} \cdot \mathbf{q}^T}{\|\mathbf{q}\|}$$

with τ a tortuosity, D_m molecular diffusion coefficient, α_L, α_T the longitudinal and transversal dispersivities. \mathbf{I} is an identity tensor.

Table 6 Example properties for tracer transport

<i>Component properties</i>	
Component name	CONCENTRATION1
Molecular diffusion coefficient	1E-8 m ² /s
<i>Medium properties</i>	
Tortuosity	1.0
Longitudinal dispersivity	1.0 m
Transversal dispersivity	1.0 m

As an example, we consider transport properties shown in Table 6. To add tracer transport modeling using GINA,

- Add new process
 - Go to “FEM-Parameter” menu and click “Process ...”.
 - Activate Chemical and enter 1 to “Number of Components”
 - Click “Save...” and “Exit”.
- Set Component Property
 - Click “Component Properties...”
 - Copy and paste the following

```
#COMPONENT_PROPERTIES
$NAME
  CONCENTRATION1
$MOBILE
  1; MOBIL-Flag: 0=immobile, 1=mobile/transported
$DIFFUSION
  1 1.0e-8 ; diffusion model type, diffusion constant
#STOP
```

-
- Set I.C.
 - Click “Initial Conditions ...”
 - Double-click an empty third row in a table.
 - Select the following options in Initial Conditions dialog
 - Process: MASS_TRANSPORT
 - Variable: CONCENTRATION1
 - Geo_type: DOMAIN

- Dis-Type: CONSTANT, 0
- Click "Apply" and "Exit"
- Click "Save..." and "Exit"
- Set B.C.
 - Click "Boundary Conditions ..."
 - Double-click an empty second row in a table.
 - Select the following options in Initial Conditions dialog
 - Process: MASS_TRANSPORT
 - Variable: CONCENTRATION1
 - Geo_type: POINT, INJECTION
 - Dis-Type: CONSTANT, 1.0
 - Click "Apply" and "Exit"
 - Click "Save..." and "Exit"
 - Click "Save" and "Exit".
- Set Numerical Property
 - Click "Numerical Properties..." to open "Edit" dialong
 - Add the following above #STOP and save.

```
#NUMERICS
$PCS_TYPE
MASS_TRANSPORT
$LINEAR_SOLVER
; method error_tolerance max_iterations theta precondition storage
2 6 1.e-010 3000 1.0 1 4
$NON_LINEAR_ITERATIONS
PICARD LMAX 1 0.0 1e-3
; type -- error_method -- max_iterations -- relaxation -- tolerance(s)
$COUPLING_CONTROL
; error method -- tolerances
LMAX 1.e-3
```

-
- Set Output Parameter
 - Click "Output Parameters ..."
 - Double-click the first row in a table. The first row has settings for outputting entire domain results into PVD/VTU files.
 - In Output dialog, activate CONCENTRATION1 in Node_Values. Apply and exit.
 - Double-click the second row. The row has settings for outputting results at the production point.
 - In Output dialog, activate CONCENTRATION1 in Node_Values. Apply and exit.
 - Save and Exit.

Remark

- Medium properties for mass transport such as tortuosity, longitudinal and transversal dispersivities are already defined in section 4.6.3.

7.6 Time-dependent boundary condition

Users can give time-dependent boundary conditions in BC and ST files. To do it, there are two steps users should follow.

First, define time-series data (time, value) used for boundary conditions in a RFD file (e.g. geothermal.rfd). Definition of time series data begins with keyword `#CURVES`. Users can have multiple time-series data in a RFD file. The order of the definitions is used as a curve id in BC or ST files (Note: numbering starts from 1). The below example defines a curve which has 1.0 from time 0s to 86400s and becomes zero afterward, i.e. pulse injection.

```
#CURVES ; Curve 1
  0 1.0
  86400 1.0
  86401 .0
#STOP
```

Next, configure BC or ST files to use the defined curve. Differences in those files from that for temporally constant boundary conditions are using `CONSTANT 1` to `$DIS_TYPE` and adding the new keyword `$TIME_TYPE`. For example, to set time-dependent concentration at the injection using the above curve, BC for mass transport in geothermal.bc should be modified as below. A number after `CURVE`, e.g. 1, indicates a curve id defined in RFD files. As RFD files can have multiple curves, users should specify which curve is used for the boundary condition.

```
#BOUNDARY_CONDITION
$PCS_TYPE
  MASS_TRANSPORT
$PRIMARY_VARIABLE
  CONCENTRATION1
$GEO_TYPE
  POINT INJECTION
$DIS_TYPE
  CONSTANT 1
$TIM_TYPE
  CURVE 1
#STOP
```

7.7 Importing/exporting simulation results as initial conditions

Setting appropriate initial conditions is import for any transient analysis. For example, in the above tutorial, we assume in-situ pore-pressure follows hydrostatic and give explicit formula in IC files. However, giving explicit formula is difficult in some cases such as when regional groundwater flow is not negligible or when fluid density varies with temperature. In those cases, OpenGeoSys offers a two-step simulation approach: pre-computing and main-computing. First, users run a

simulation to get in-situ reservoir conditions, i.e. steady state (pre-computing). Next, using pre-computing results as initial conditions, users can start a transient simulation (main-computing). Detail procedures are following,

- Prepare input files for pre-computing.
 - In PCS files, add \$RELOAD keyword with an option “1 1” to write results into files (*.asc) which can be later used as initial conditions. An example for hydraulic process is given below.

```
#PROCESS
$PCS_TYPE
  LIQUID_FLOW
$RELOAD
  1 1
#STOP
```

- Remark that simulations should include at least two time steps.
- Run a simulation.
- Computed results will be written in ASC files, e.g. geothermal_LIQUID_FLOW_PRESSURE1_primary_value.asc.
- Prepare input files for main-computing.
 - Copy the generated ASC file to a directory having input files for main-computing.
 - In PCS files, add \$RELOAD keyword with an option “2” to make OpenGeoSys import the ASC file as initial conditions. An example for hydrothermal process is given below.

```
#PROCESS
$PCS_TYPE
  LIQUID_FLOW
$RELOAD
  2
#PROCESS
$PCS_TYPE
  HEAT_TRANSPORT
#STOP
```

- Run a simulation.
- Plot results at time 0 and check if initial conditions are correctly applied.