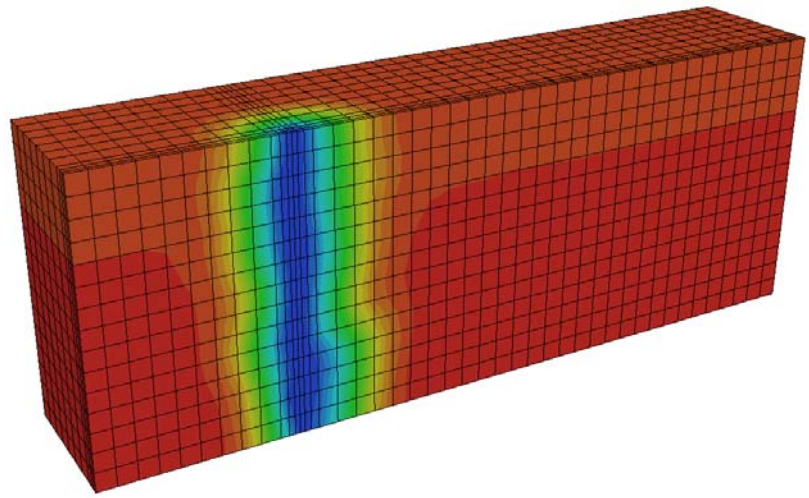


3D Geothermal Reservoir Modelling
Case Study: Urach Spa

Step by Step Tutorial

Norihiro Watanabe[1, 2], Herbert Kunz[3], 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)

GeoSys – Preprint
Version 0.5 [2009-06]

Leipzig, June 2009

Table of Contents

1 Introduction 3
1.1 Urach Spa geothermal research site 3
1.2 Conceptual model 3
1.3 References 4
2 Software Installation 5
2.1 Getting software 5
2.2 GeoSys/Rockflow 5
2.3 GINA4 5
2.4 Meshing tools (Gmsh) 5
2.5 Visualization tool (ParaView) 5
3 Start-up GeoSys Project 6
4 Geometrical model and Mesh generation 8
4.1 Start GINA4 8
4.2 Geometrical model (Half domain) 8
4.2.1 Create points 8
4.2.2 Creating polylines 9
4.2.3 Creating surfaces 10
4.2.4 Navigating 3D model with Mouse 12
4.3 Mesh generation (Hexahedral elements) 13
4.3.1 Prepare a geometry for the top surface 13
4.3.2 Create a 2D rectangular mesh for the top surface 14
4.3.3 Extrude 2D-Elements to 3D Elements 21
4.4 Adding the GLI file and MSH file to the GeoSys project 23
5 Simulation of Hydraulic process (H) 26
5.1 Creating LIQUID_FLOW process 29
5.2 Time discretization 30
5.3 Numerical properties 31
5.4 Output properties 32
5.5 Initial Conditions 34
5.6 Boundary Conditions 35
5.7 Material properties (MFP/MSP/MMP) 39
5.8 Run simulation 44
5.9 Visualization of simulation result with ParaView 45
6 Simulation of heat transport (T-H coupled processes) 50
6.1 Creating HEAT_TRANSPORT process 50
6.2 Time Discretization 51
6.3 Numerical properties 51
6.4 Data output 52
6.5 Initial Conditions 53
6.6 Boundary Conditions 54
6.7 Fluid properties (MFP) 54
6.8 Solid properties (MSP) 54
6.9 Porous medium properties (MMP) 56
6.10 Run T simulation 56
6.11 Coupling H and T Processes 57
6.12 Visualize results 60

1 Introduction

This step-by-step tutorial shows you how to setup a three dimensional geothermal reservoir model through a case study of the Urach Spa geothermal site. The reservoir is represented as homogeneous porous media.

1.1 Urach Spa geothermal research site

The Urach Spa geothermal research site is approximately 50 km southeast of Stuttgart, southern Germany, at the northern boundary of the Jurassic Swabian Alb. It is located in a very dense gneiss formation, in an area that is almost tectonically inactive, in the center of a large geothermal anomaly with a gradient of up to 110C/km within the first 300 m depth. Below, the gradient decreases to 40C/km down to 1600 m. At greater depths, within the crystalline basement, the geothermal gradient is about 30C/km, which is similar to the average geothermal gradient of the earth.

1.2 Conceptual model

The reservoir depth is between 3850m-4150m. The proposed boreholes (U3 and U4) dipole flow circulation system (i.e. a "doublet") are located 400m apart. Based on the large amount of scientific data available on the Urach Spa reservoir, a three-dimensional model of the reservoir system is developed. The hydraulically active areas allowing the reservoir to be represented geometrically as a cuboid are 300m high, 300m wide and 800m long (Figure 1). Measured transmissibility of the reservoir is 0.3 Dm. The apparent homogeneous permeability over the roughly 300m thick layer is:

- $k = \frac{T_m}{z} = \frac{0.3\,Dm}{300\,m} \approx 10^{-15} m^2$

Concerning the initial conditions ($t = 0$), we assume linear depth-dependent hydrostatic pressure and temperature distribution. The geothermal gradient according to the temperature logs in the reservoir depth range of U3 is ω = 0.3K/m:

- $T(t=0) = 435.15 + \omega(-4445.0 - z)$ [K]

The injection well is considered to have an overpressure of 10MPa and the production well an underpressure of 10MPa. Fluid injection temperature is assumed to be 50°C.

This tutorial assumes constant fluid properties in the reservoir, although fluid properties are actually non-linear functions of temperature, pressure and salinity.

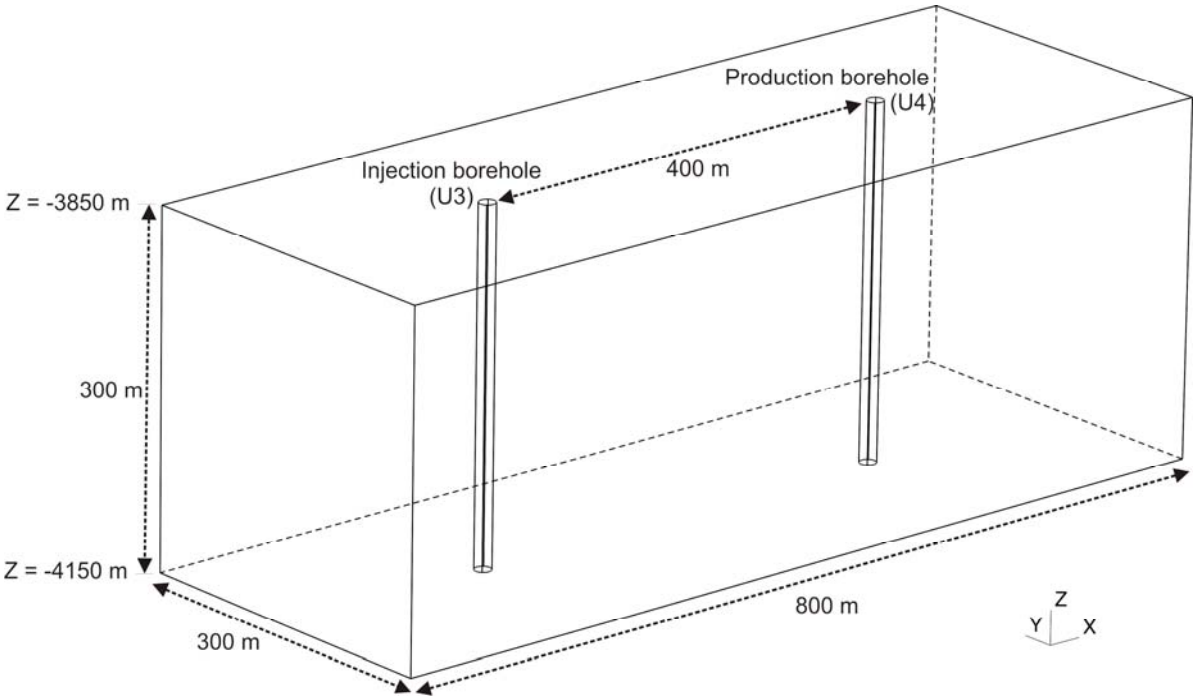


Figure 1. Cuboid reservoir model with a borehole doublet (U3 and U4)

1.3 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, Randriamanjatosoa 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.

Tenzer H, McDermott CI, Kolditz O (2009): Comparison of the exploration and evaluation of enhanced HDR geothermal sites at Soultz-sous-Forêts and Urach Spa. Environmental Earth Sciences, submitted.

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.

2 Software Installation

2.1 Getting software

We prepare an installation CD which includes all necessary software. Insert the CD into your computer and copy files on your hard disk.

Programs to be installed are following,

- GeoSys: Simulation software with GUI
- GINA4: Pre-Post processing software for Rockflow/GeoSys
- Gmsh: Free meshing tool
- ParaView3: Free 3D Visualization tool

2.2 GeoSys/Rockflow

The following files are necessary to run GS/RF-GUI:

GeoSys.exe	Windows application
shapelib.dll, fdelaun2d.dll, fdelaun3d.dll	Additional libraries

These files should be whether altogether in one directory or in the system directory.

New: use GeoSysGUI.msi for installation.

2.3 GINA4

GINA4.EXE	Windows application
comdlg32.ocx, msflxgrd.ocx, richtx32.ocx, comctl32.ocx, mscomctl.ocx, msvbvm60.dll	System files (Visual Basic 6.0 Runtime library)

2.4 Meshing tools (Gmsh)

For meshing, Gmsh (http://www.geuz.org/gmsh/) can be used for triangulation. The following files should be in the same directory as GeoSys.exe is.

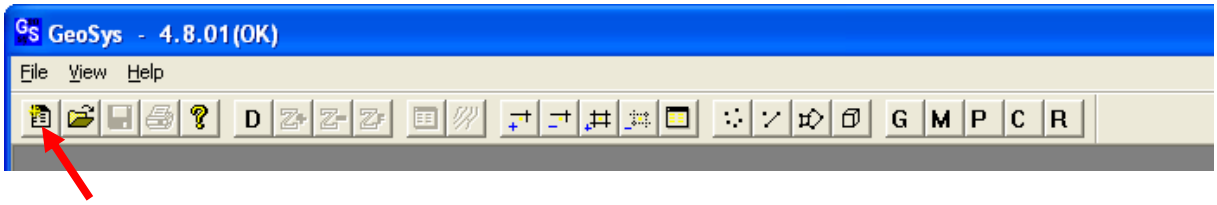
Gmsh.exe	Windows application
cygwin1.dll	Cygwin library

2.5 Visualization tool (ParaView)

If ParaView has not been installed on your computer yet, run ParaView installation file, e.g. "*paraview-3.2.1-win32-x86.exe*", and follow instructions. You can also download ParaView from: http://www.paraview.org/

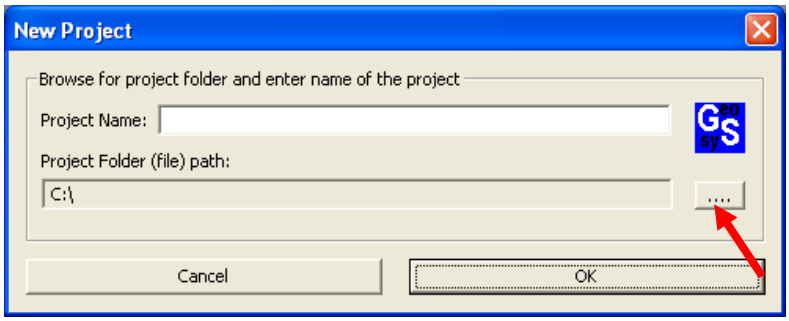
3 Start-up GeoSys Project

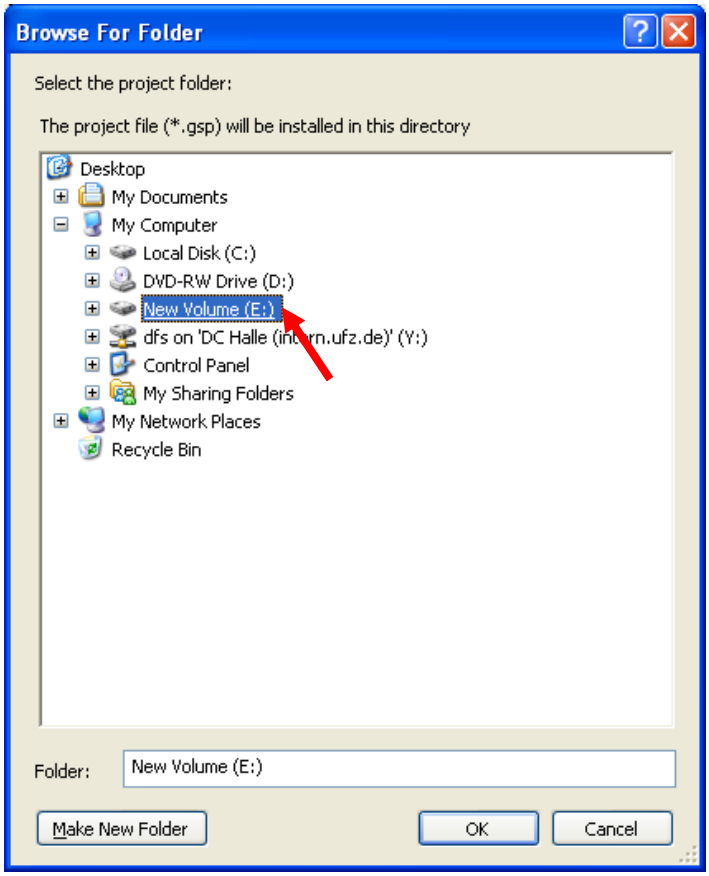
To create a new GeoSys project, use either the File option from the tool bar or in



After a new project has been created (clicking OK button), a project folder will be created automatically in the directory chosen as project folder path. The name of the folder is the same as the name of the project. The steps are as follows:

- Select the base directory using the file browser

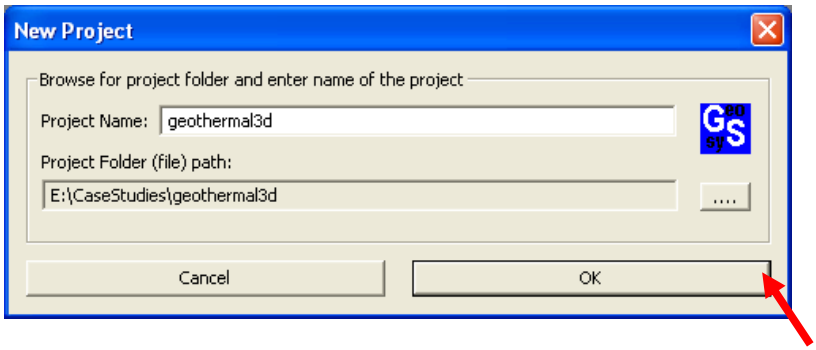




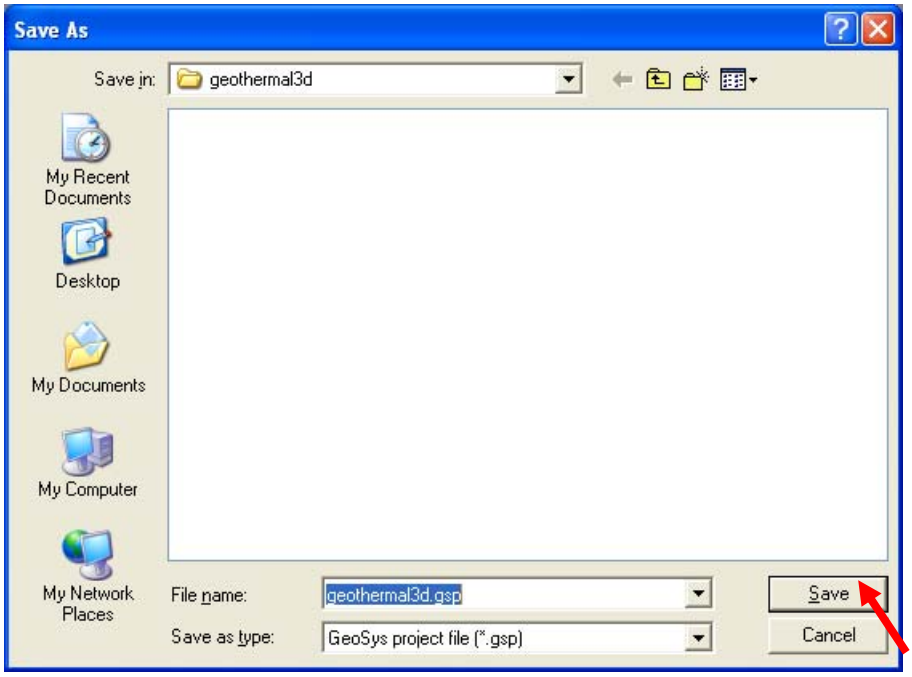
Here the base directory will be created in, for example, *E:\CaseStudies*

Enter the project name “geothermal3d” and confirm with OK

- The project directory will be *E:\CaseStudies\geothermal3d*

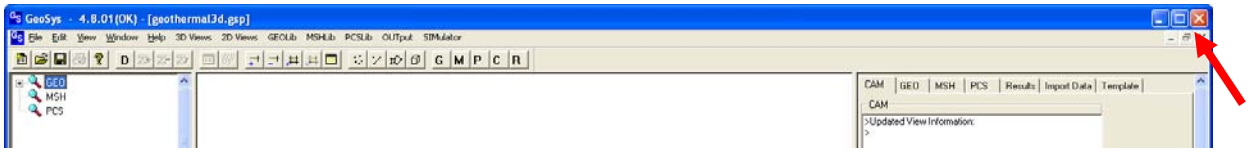


Click Save. The GeoSys project file (*.gsp) will be created in the project folder.



Congratulation! You succeeded to create a new GeoSys project. Now you can find GeoSys project files (*geothermal3d.gsp*, *geothermal3d_ascii.gsp*) on the project directory.

Close GeoSys-GUI as we will come back later.



4 Geometrical model and Mesh generation

Creating a geometrical model and a spatial discretization of a problem (a mesh data) are necessary for a finite element simulation. In this tutorial, you will create a geometrical model and a mesh for the geothermal reservoir using GINA. As the problem is symmetric about the short axis, only half the reservoir is actually simulated. The geometry you will create is shown in Figure 2.

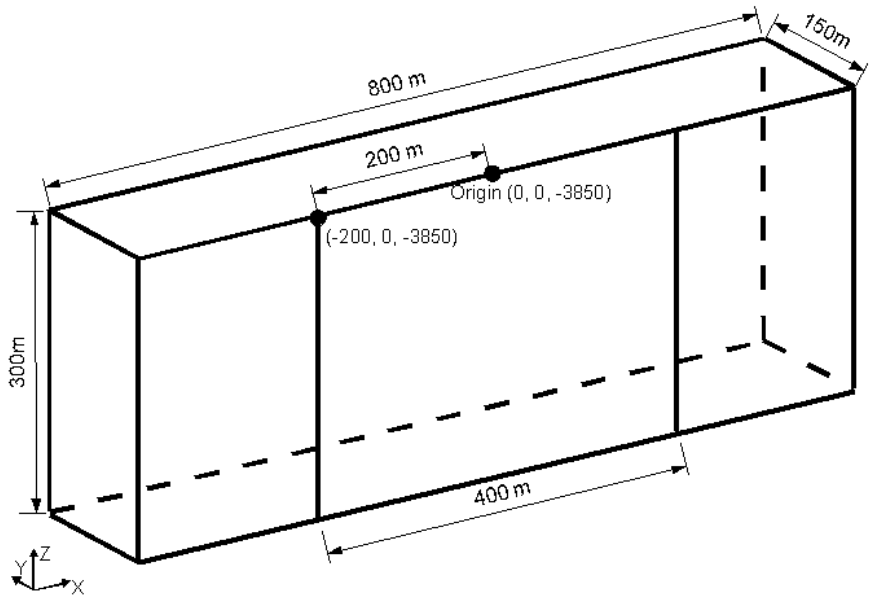


Figure 2. Geometry for half domain

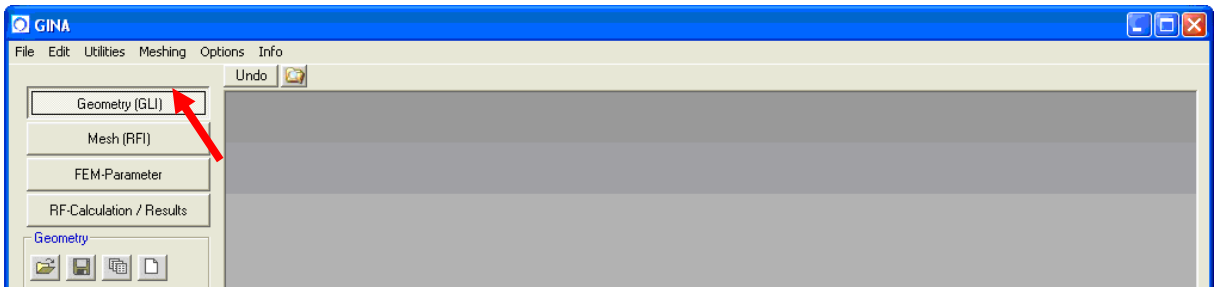
4.1 Start GINA4

Open Explorer and browse the directory where you put the GINA4 program.

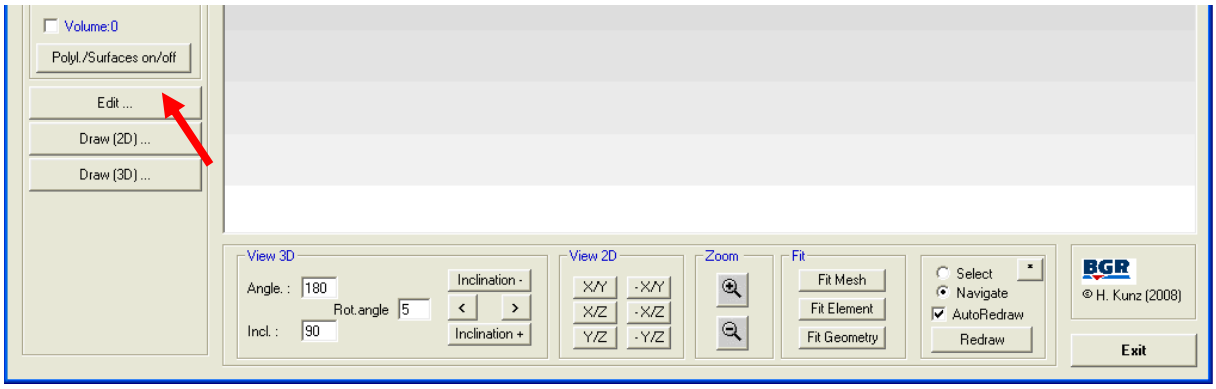
4.2 Geometrical model (Half domain)

4.2.1 Create points

Click “Geometry (GLI)” button.

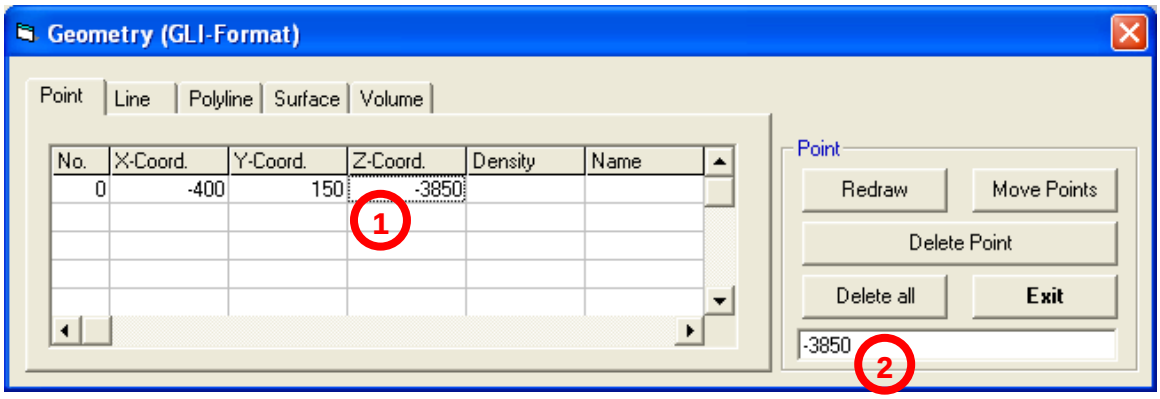


Click “Edit…” button.



Giving point information (e.g. point No., coordinates),
1. Select a cell where you want to set.
2. Enter value into the the right-side textbox.

* Notice: Please start Point No. with zero (0).



Enter coordinates for all points as below.

No.	X-Coord.	Y-Coord.	Z-Coord.
0	-400	150	-3850
1	400	150	-3850
2	400	0	-3850
3	-400	0	-3850
4	-200	0	-3850
5	200	0	-3850
6	-400	150	-4150
7	400	150	-4150
8	400	0	-4150
9	-400	0	-4150
10	-200	0	-4150
11	200	0	-4150

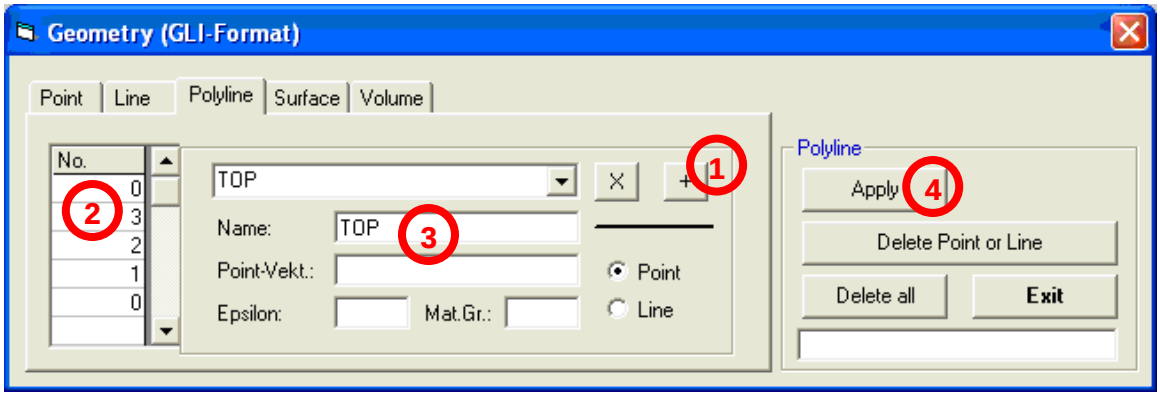
4.2.2 Creating polylines

Click “Polyline” tab on the Geometry dialog.

Create a new polyline.

1. Click "+(plus)" button to add a new polyline.
2. Enter point no. to make a polyline (e.g. 0,3,2,1,0).
3. Set name of the polyline (e.g. TOP)
4. Click "Apply"

* Notice: A closed polyline should have the same point no. at the beginning and at the end of a point list.

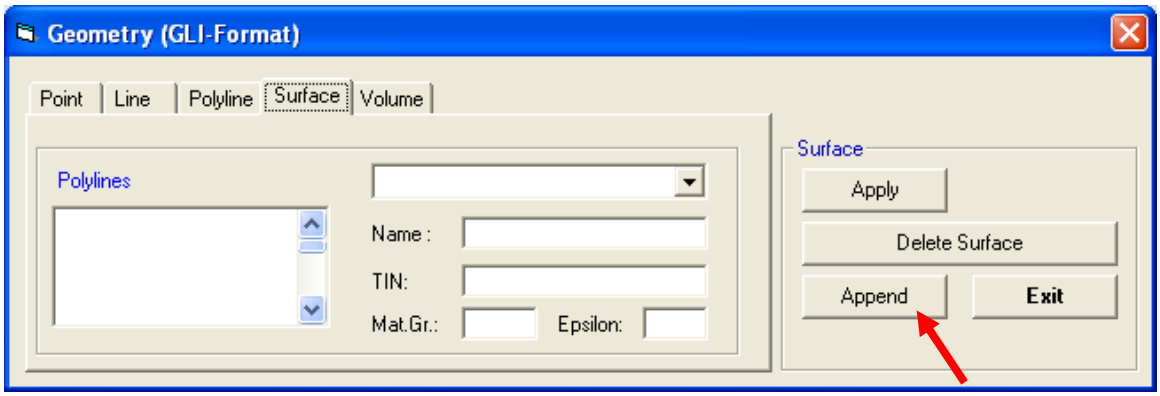


Enter the below polyline data.

Name:	Point No.
TOP	0 3 2 1 0
BOTTOM	6 7 8 9 6
RIGHT	1 2 8 7 1
LEFT	0 3 9 6 0
FRONT	3 2 8 9 3
BACK	0 1 7 6 0
INJECTION_WELL	4 10
PRODUCTION_WELL	5 11

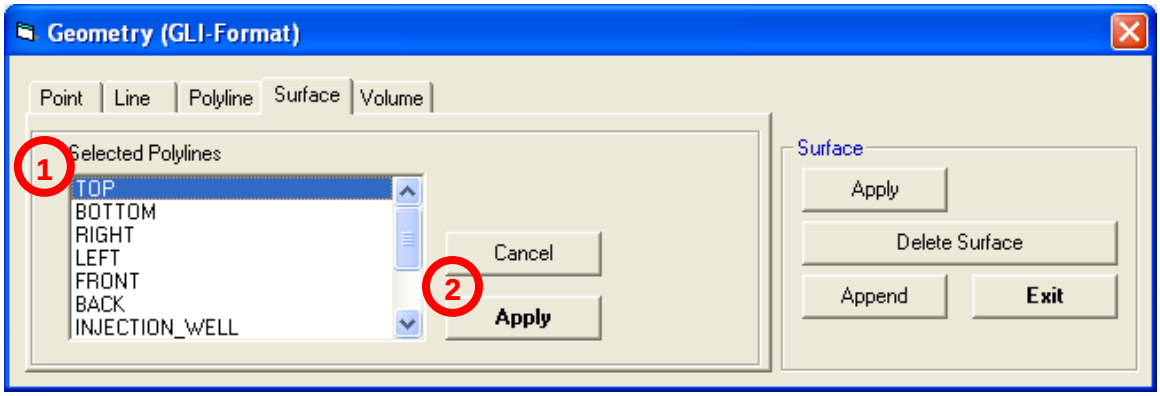
4.2.3 Creating surfaces

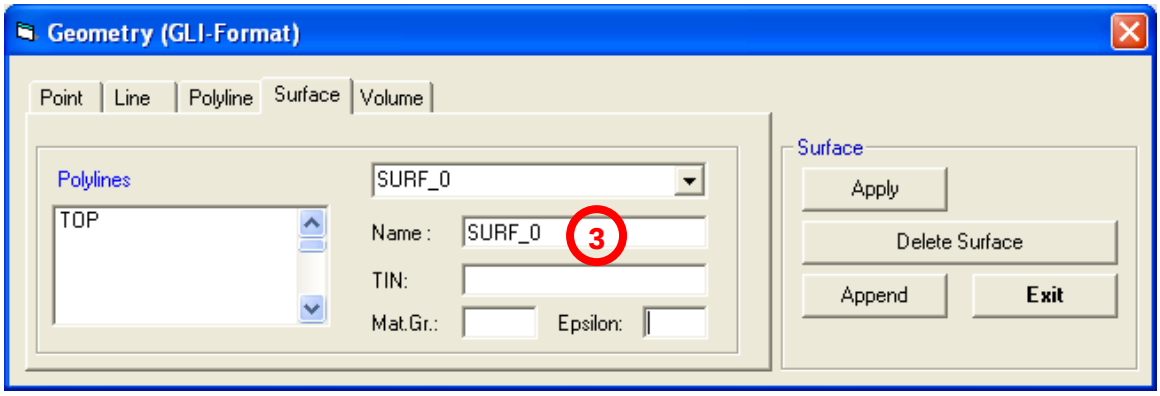
Move to "Surface" tab on the Geometry dialog and click "Append" button.



Create a new surface,

1. Select a polyline which makes a new surface (e.g. TOP).
2. Click “Apply”.
3. Enter a name of the surface (e.g. SURF_0).



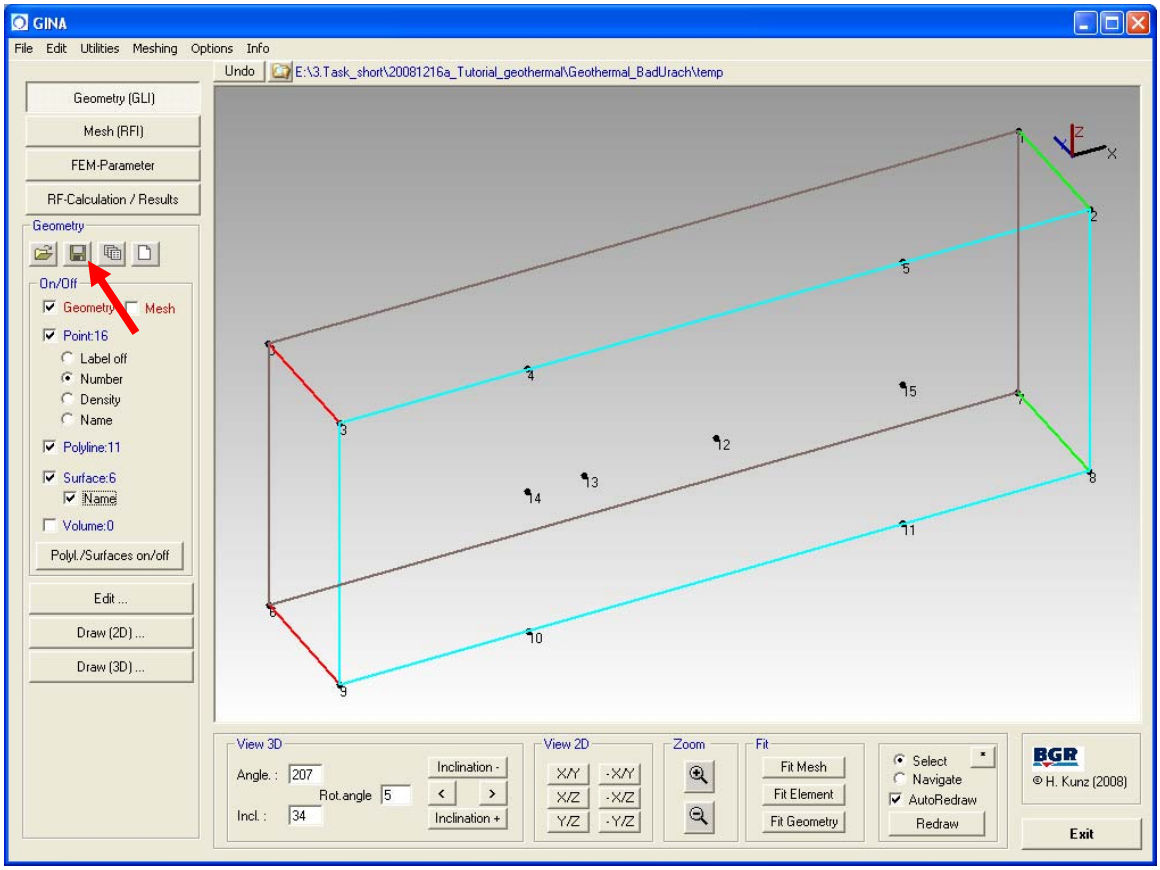


Enter the below surface data.

Surface name:	Polylines:
SURF_0	TOP
SURF_1	BOTTOM
SURF_2	RIGHT
SURF_3	LEFT
SURF_4	FRONT
SURF_5	BACK

Click “Exit” button.

Save geometry data into a GLI file. Click the “save” icon as following.

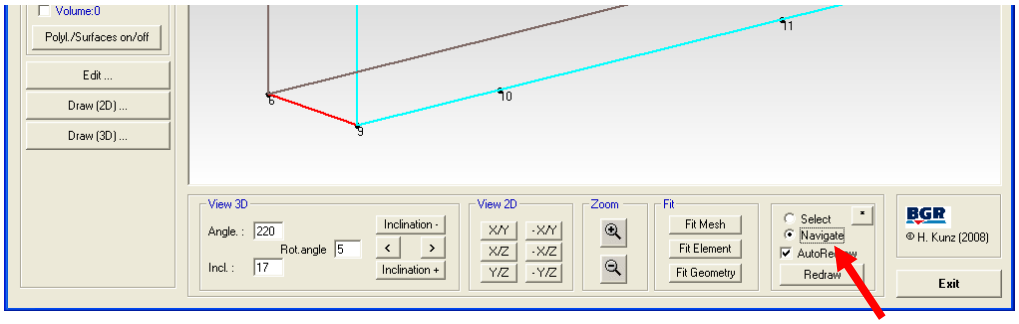


Save the geometry into file.

1. Click “Save” icon on the geometry menu.
2. Right click on Save File dialog to create a new folder, e.g. “*E:\\CaseStudies\geothermal3d\geo*”.
3. Move to the new folder and enter “geothermal3d.gli”.
4. Click “Save”.

4.2.4 Navigating 3D model with Mouse

Select “Navigate” option.



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

4.3 Mesh generation (Hexahedral elements)

Next step is spatial discretization of the problem, i.e. meshing. You will create a hexahedral mesh (Figure 3). Meshing procedure presented here includes two steps: 1) create a 2D quadrilateral mesh for the top surface and 2) extrude the mesh into the z-direction to define a 3D hexahedral mesh. Element size is basically around 20m. Elements near the boreholes have to be refined (min. 5m) due to appearance of high gradient pressure distribution.

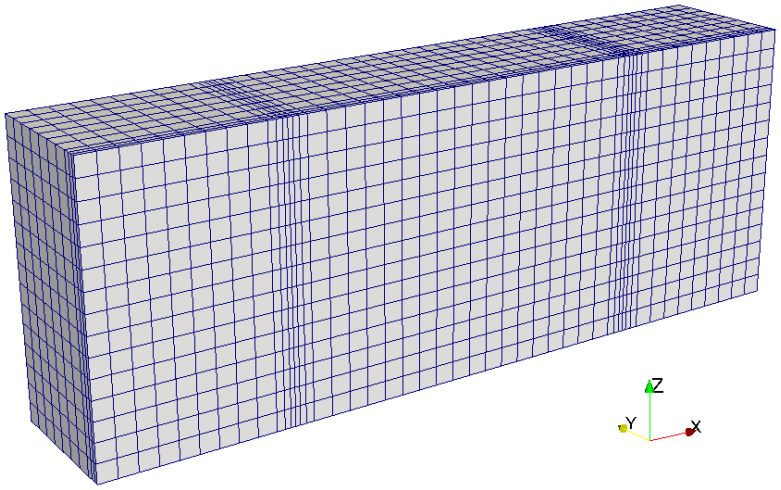


Figure 3. Hexahedral mesh

4.3.1 Prepare a geometry for the top surface

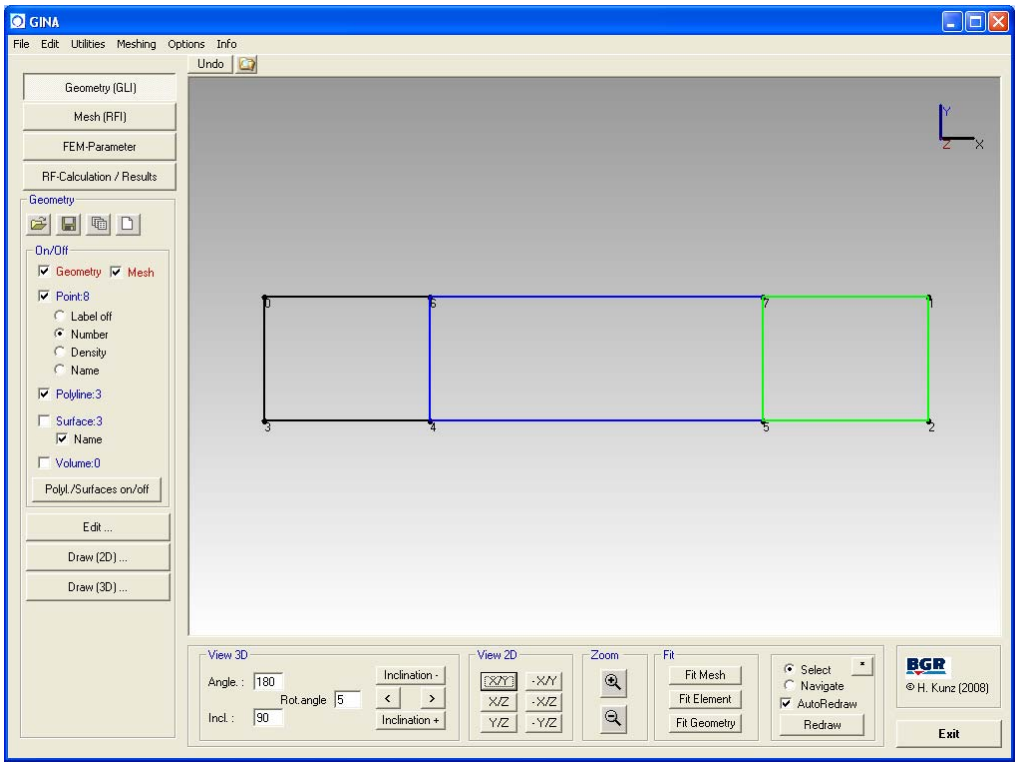
First step is to define two dimensional geometries of top surface for meshing.

Exit GINA4 and restart again. (Make sure that you saved the geometry data before exiting GINA4.)
Click “Edit…” button and enter the following points, polylines and surfaces.

No.	X	Y	Z
0	-400	150	-3850
1	400	150	-3850
2	400	0	-3850
3	-400	0	-3850
4	-200	0	-3850
5	200	0	-3850
6	-200	150	-3850
7	200	150	-3850

Name:	Point No.
PLY_0	0 3 4 6 0
PLY_1	6 4 5 7 6
PLY_2	7 5 2 1 7

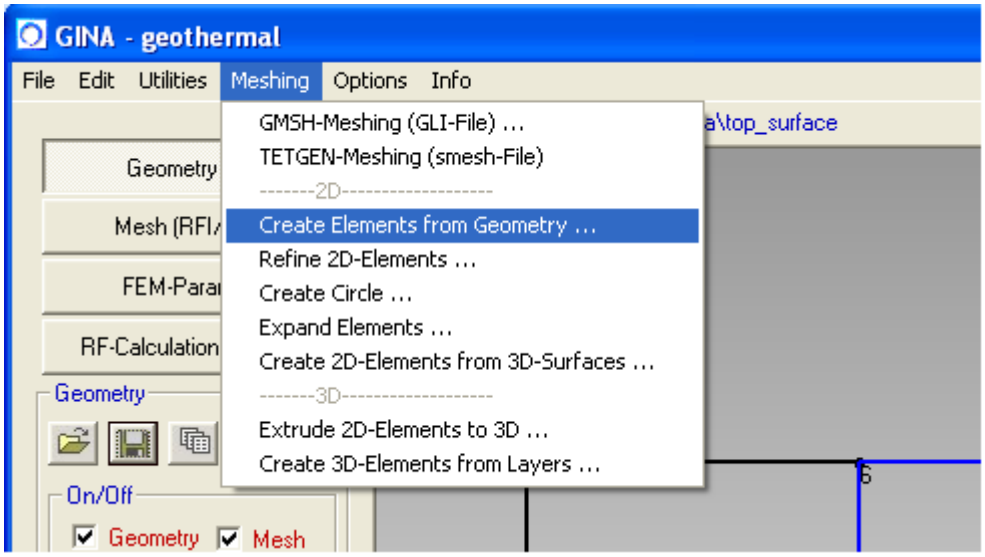
Surface name:	Polylines:
SURF_0	PLY_0
SURF_1	PLY_1
SURF_2	PLY_2



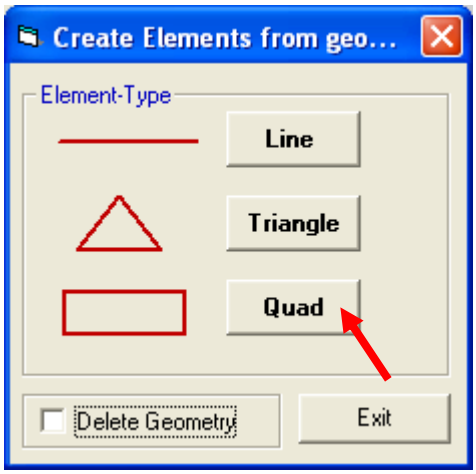
Save the geometry into a file as “top_surface.gli”.

4.3.2 Create a 2D rectangular mesh for the top surface

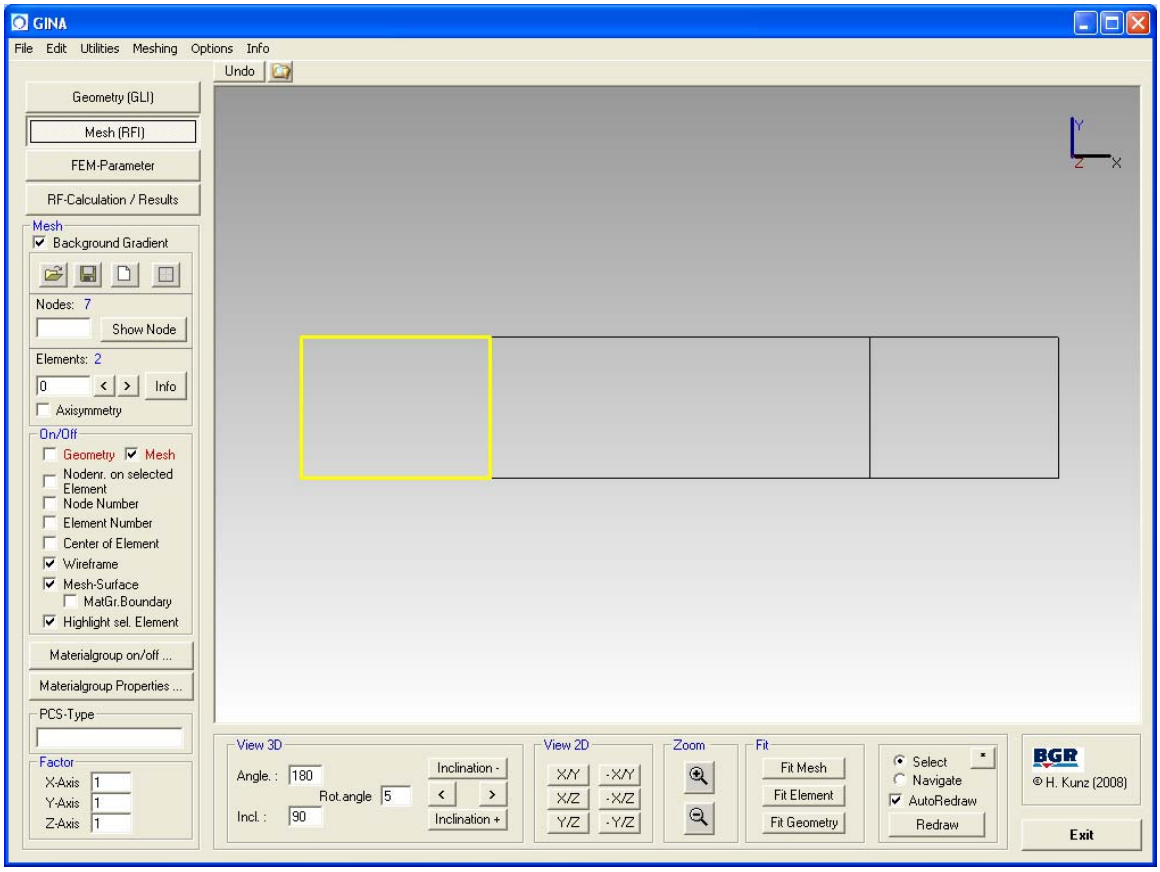
Click the “Meshing” menu > “Create Elements from Geometry…”.



Check off “Delete Geometry” and click “Quad” button. Click “Exit”.

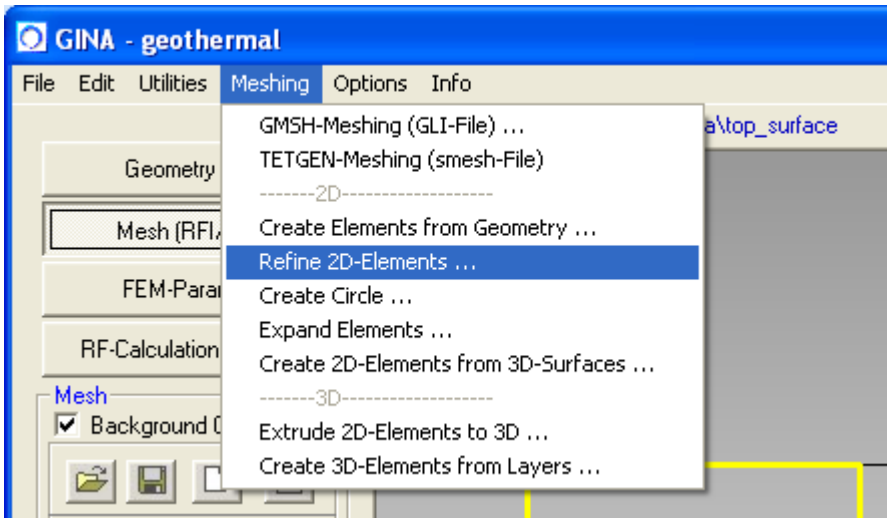


Check off “Geometry” on the Geometry (GLI) menu. Move to the Mesh (RFI/MSH) menu. Now three rectangular elements have been created.



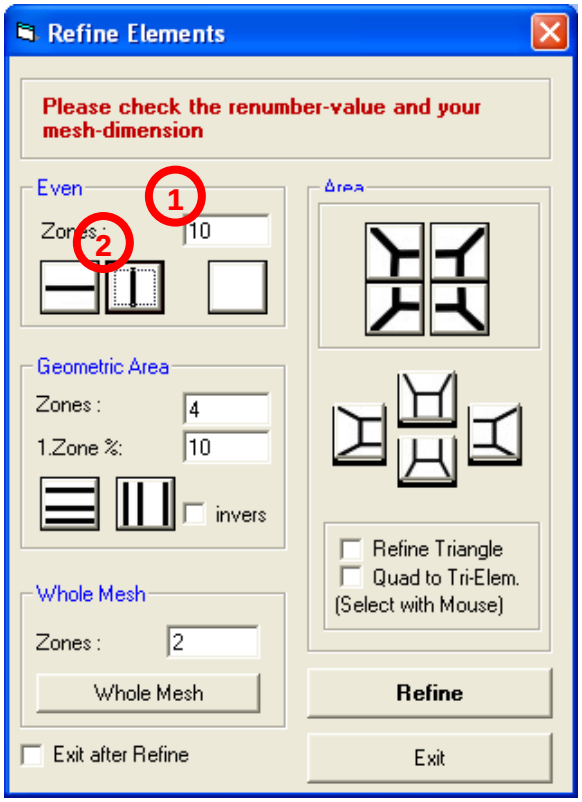
Next, the above elements are refined. Normal element size is around 20m. Elements around the boreholes are more refined.

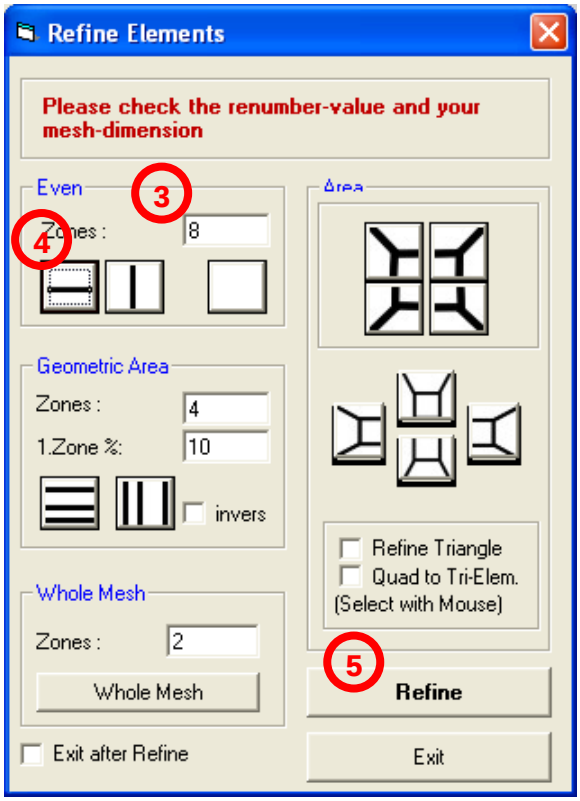
Click the “Meshing” menu > “Refine 2D-Elements …”.



Refine the left part

1. Here we focus on “Even” refinement.
2. Enter number of zones to be divided, 10.
3. Click “vertical line” button to refine vertically.
4. Enter 8 to zones.
5. Click “horizontal line” button.
6. Click “Refine” button to complete refinement.

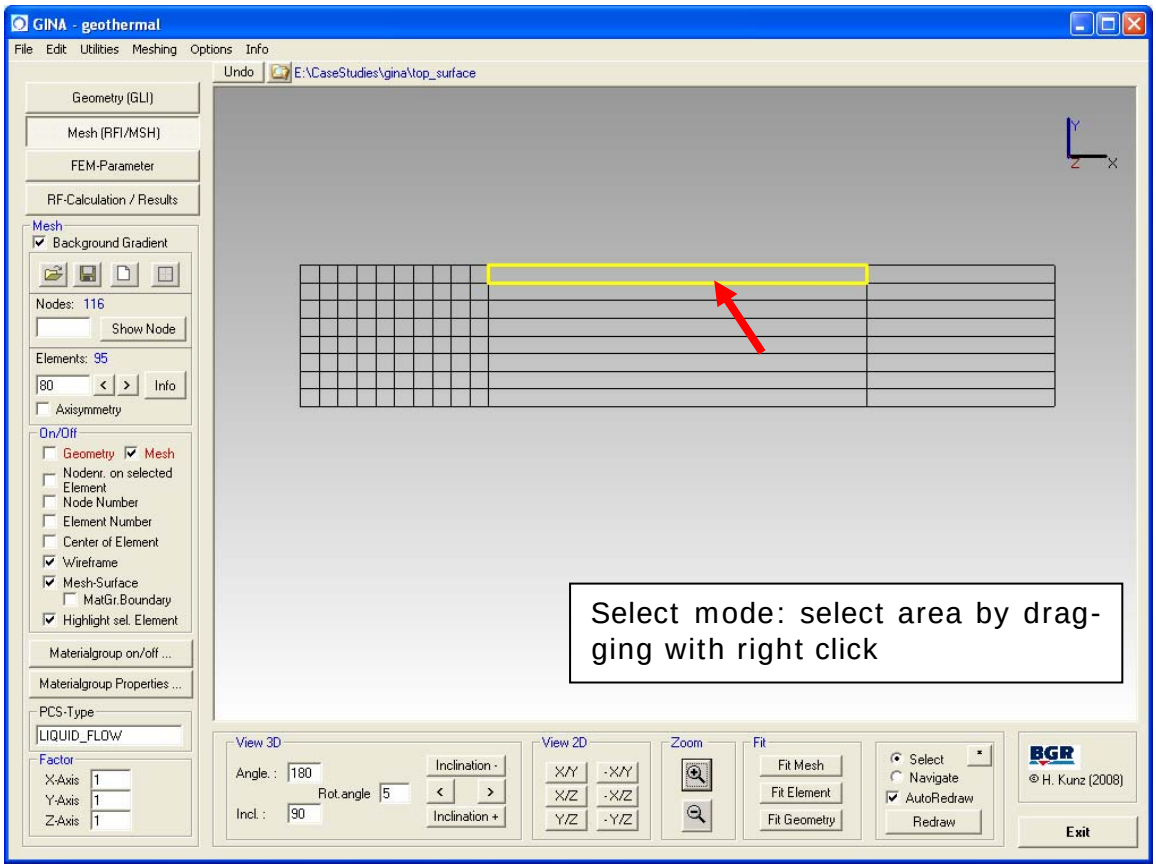


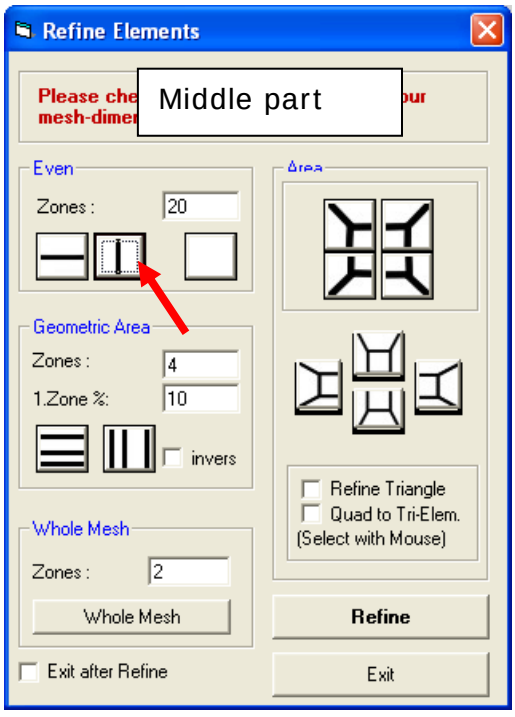


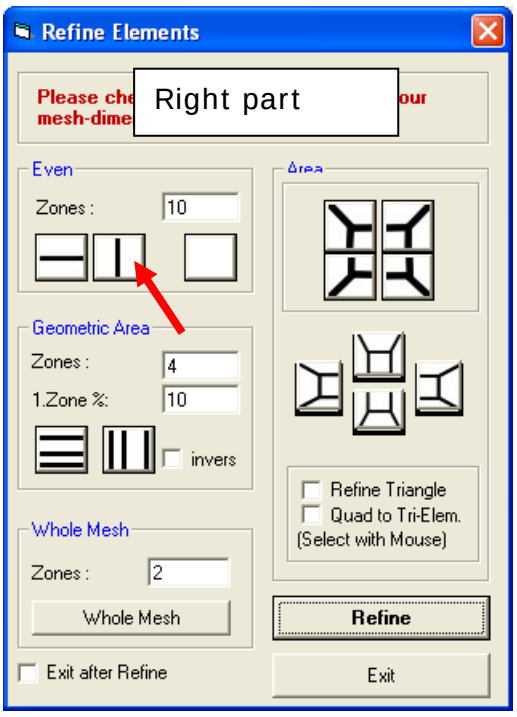
Refine the middle part and right part.

1. Turn on “Select” mode.
2. Select area which includes a element in the middle part by dragging with mouse-right click.

3. Selected elements will be highlighted.
4. Open “Refine 2D-Elements …” dialog and refine as illustrated in the below figure.
5. Refine the right part as well.

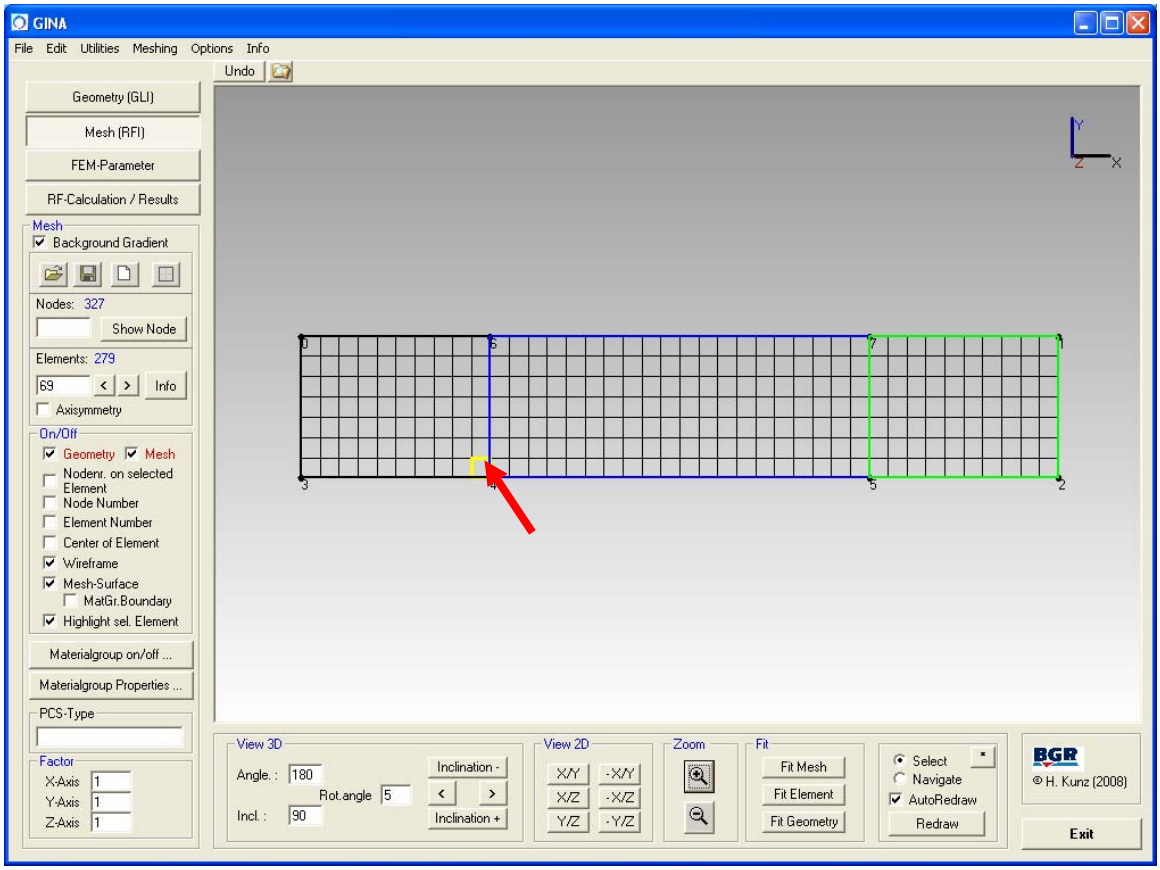


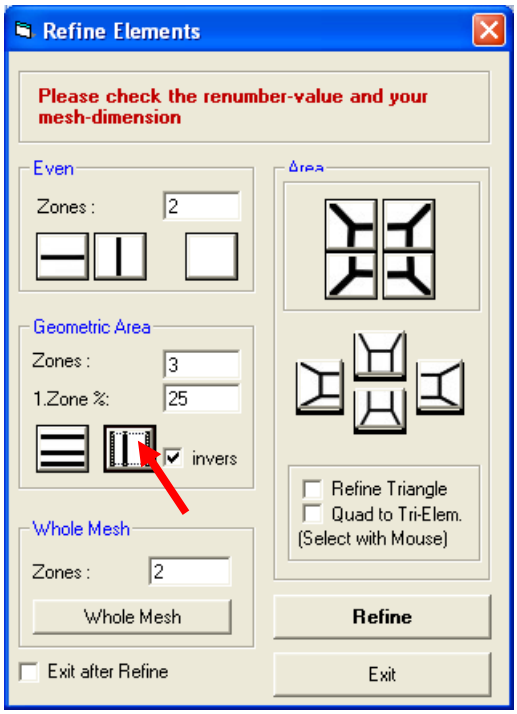


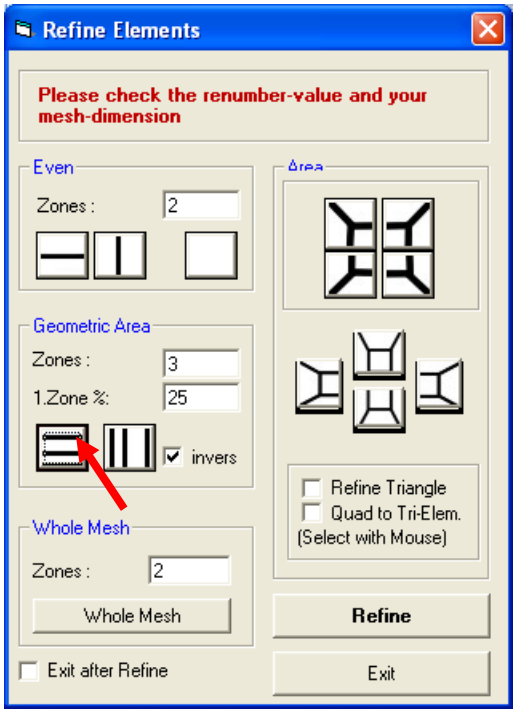


Refine elements near boreholes.

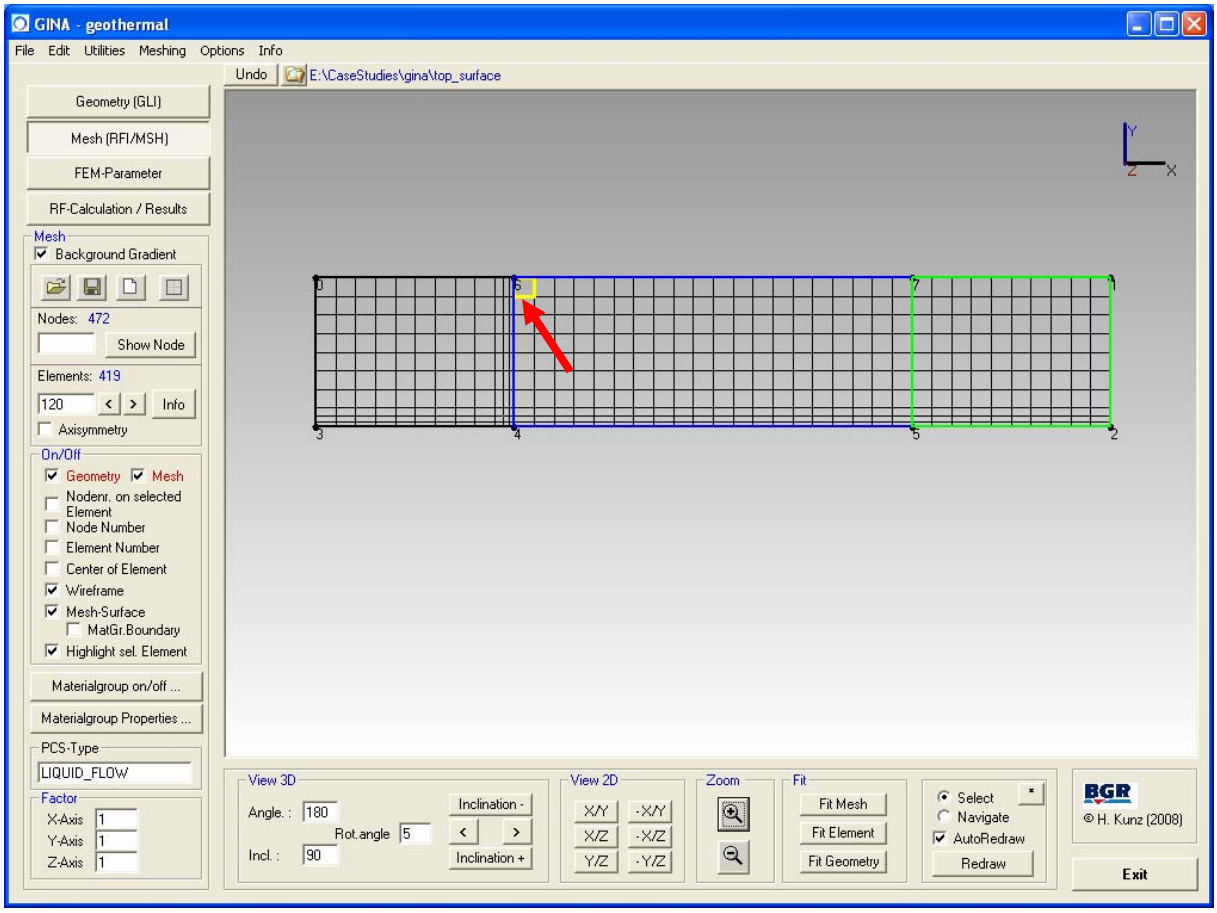
1. Select an element as shown in the below figure.
2. Open “Refine 2D-Elements …” dialog.
3. Go to “Geometric Area”.
4. Check ON “invers”
5. Enter 3 to Zones.
6. Enter 25% to 1st Zone, i.e. size of new 1st zone is around 5m.
7. Click “vertical lines” icon.
8. Click “horizontal lines” icon with the same setting.
9. Click “Refine” button.

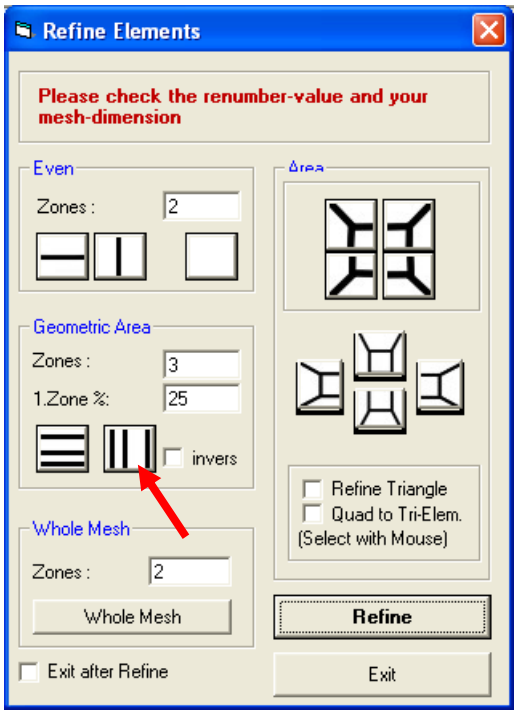


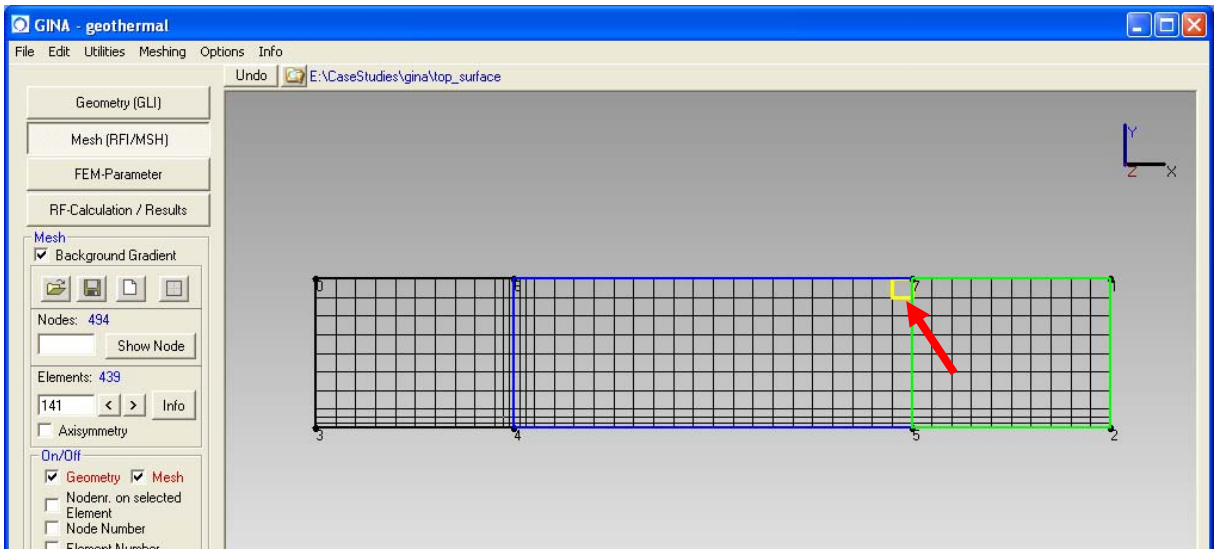


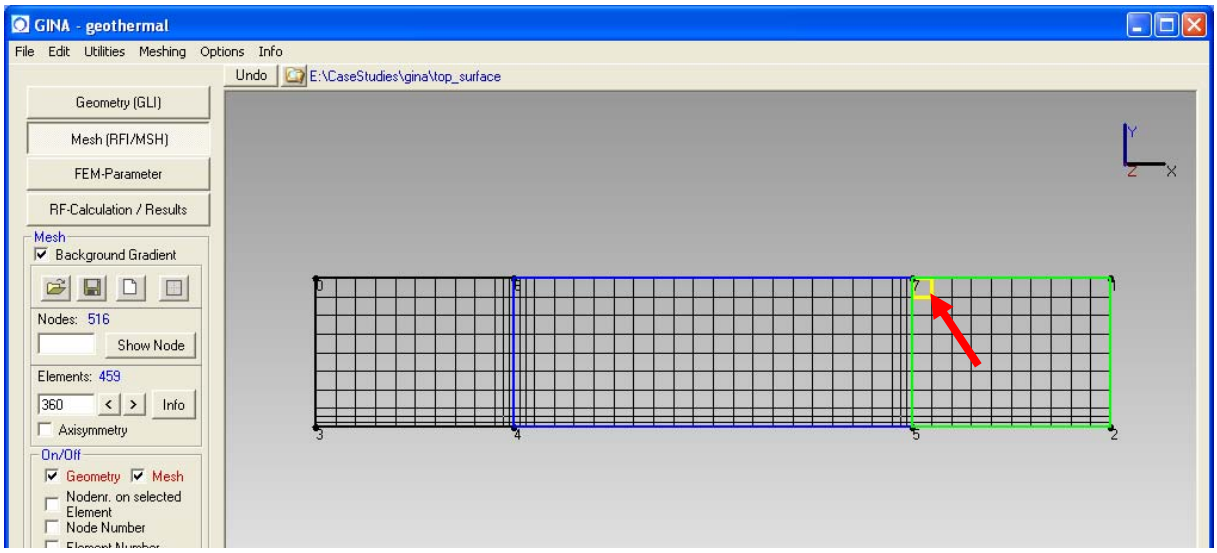


Refine other elements near the boreholes as well.

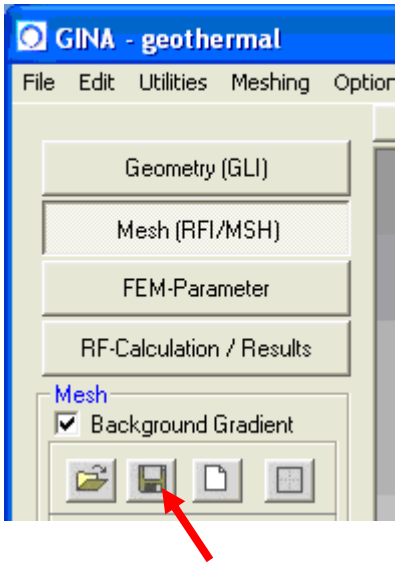



Refine Elements
Please check the renumber-value and your mesh-dimension
Even
Zones : 2
Area
Geometric Area
Zones : 3
1.Zone %: 25
invers
Refine Triangle
Quad to Tri-Elem.
(Select with Mouse)
Whole Mesh
Zones : 2
Whole Mesh
Refine
Exit after Refine
Exit


GINA - geothermal
File Edit Utilities Meshing Options Info
Undo
E:\CaseStudies\gina\top_surface
Geometry (GLI)
Mesh (RFI/MSH)
FEM-Parameter
RF-Calculation / Results
Mesh
Background Gradient
Nodes: 494
Show Node
Elements: 439
141
Info
Axisymmetry
On/Off
Geometry
Mesh
Nodenr. on selected Element
Node Number

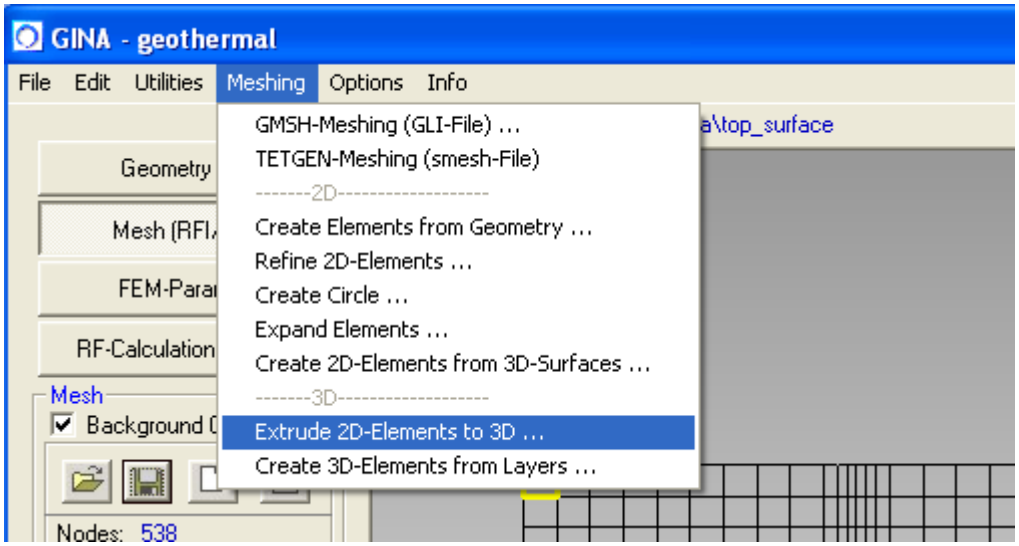

GINA - geothermal
File Edit Utilities Meshing Options Info
Undo
E:\CaseStudies\gina\top_surface
Geometry (GLI)
Mesh (RFI/MSH)
FEM-Parameter
RF-Calculation / Results
Mesh
Background Gradient
Nodes: 516
Show Node
Elements: 459
360
Info
Axisymmetry
On/Off
Geometry
Mesh
Nodenr. on selected Element
Node Number

Mesh refinements are finished. Save a mesh file as “top_surface.msh”.

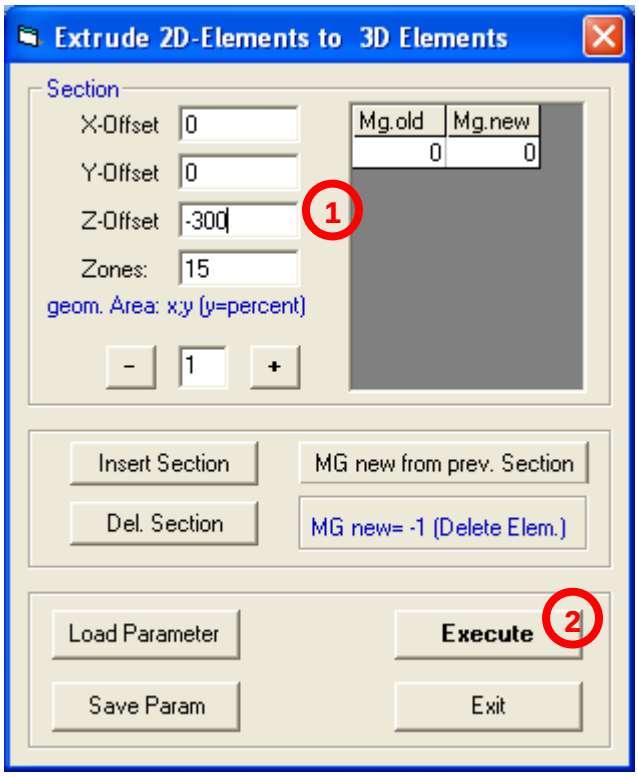


4.3.3 Extrude 2D-Elements to 3D Elements

Click the “Meshing” menu > “Extrude 2D-Elements to 3D …”.

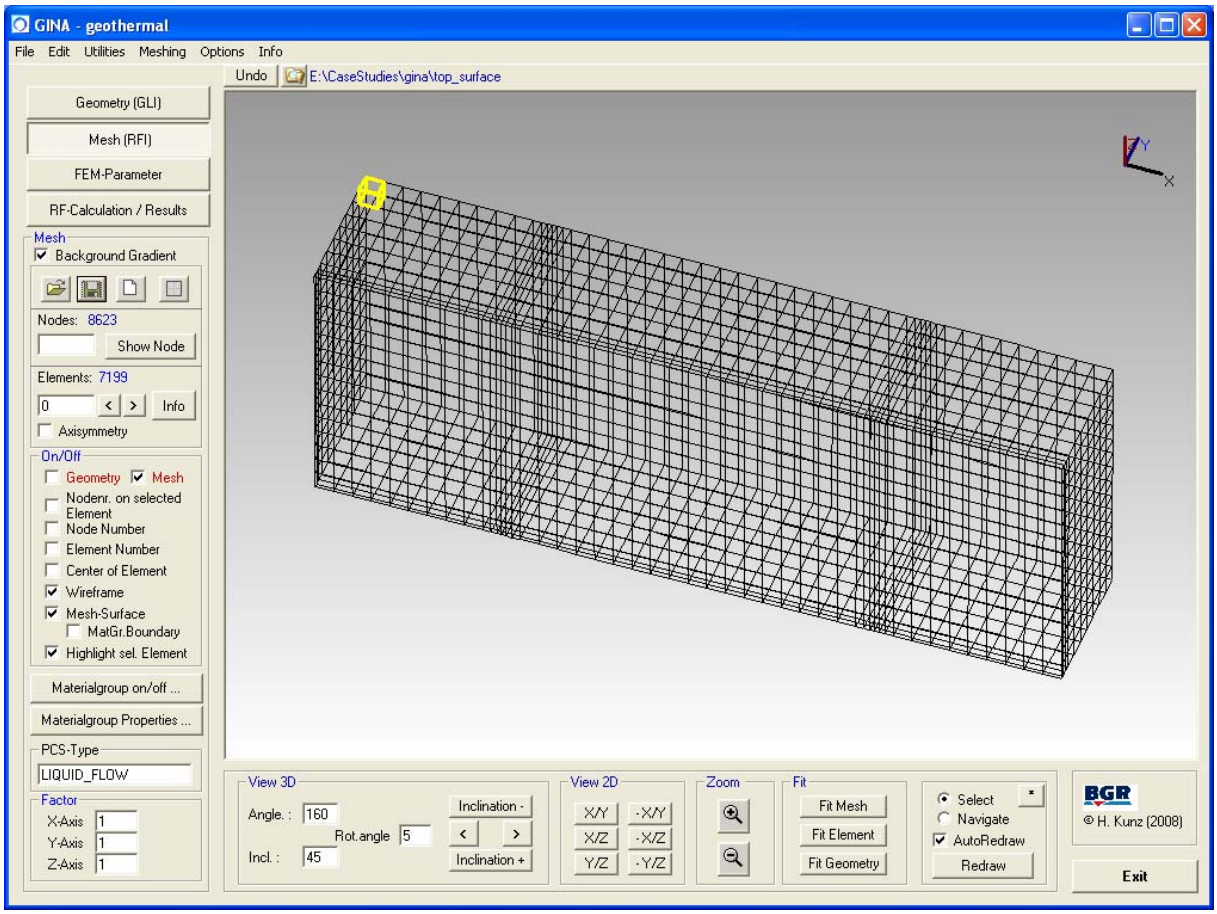


Enter -300 to Z-Offset and 15 to Zones. Click “Execute”.



Click “Exit” to close the dialog.

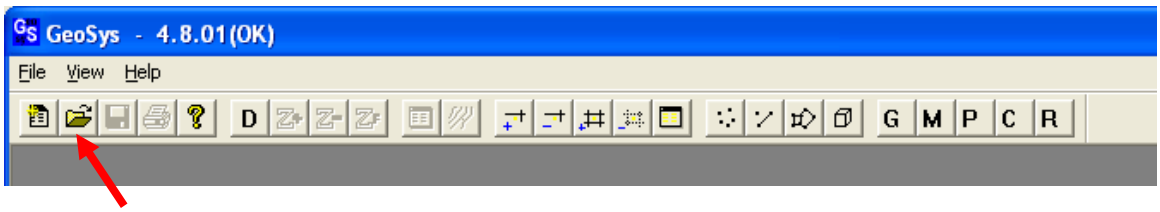
Save the created 3D mesh into a file. Mesh file name will be “geother-mal3d.msh”.



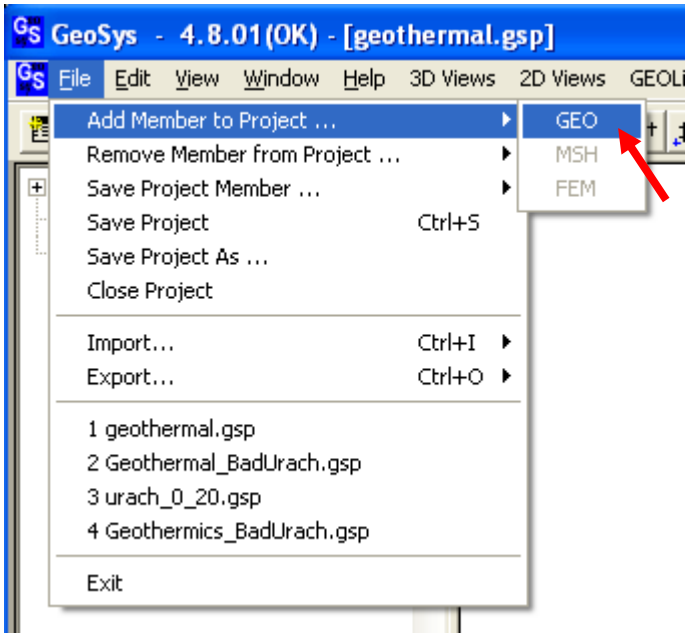
Exit GINA4.

4.4 Adding the GLI file and MSH file to the GeoSys project

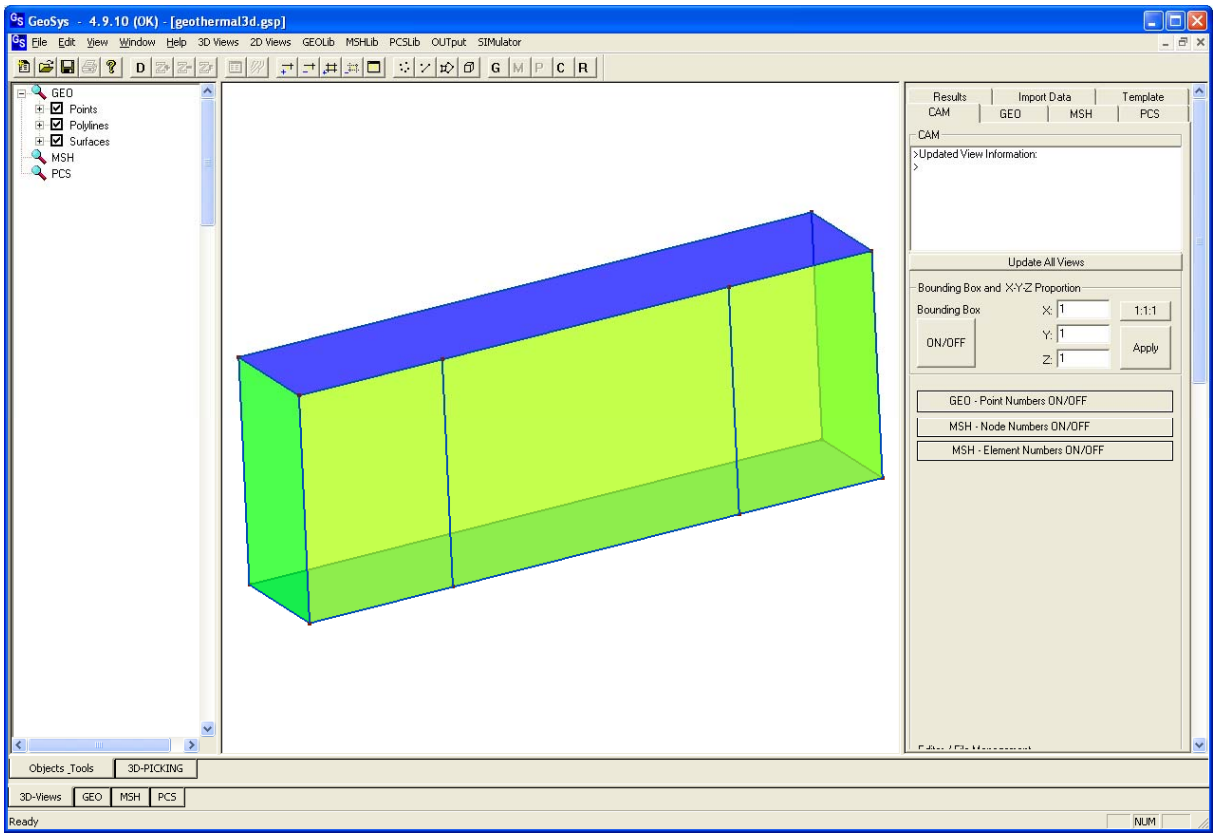
Start GeoSys and open the GeoSys project “geothermal3d.gsp” we created before.



Add the geometry file to the project. Click “File” menu > “Add Member to Project …” > “GEO”. Select the geometry file “geothermal3d.gli” you created with Gina.

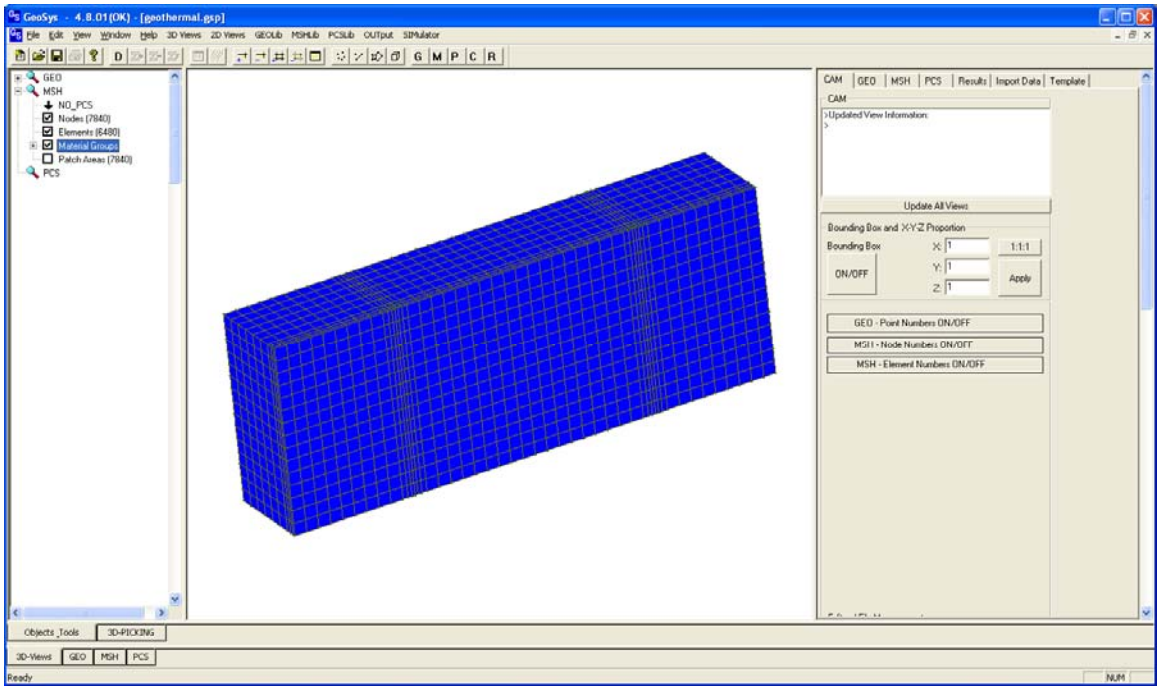


Imported geometries will be displayed as below.

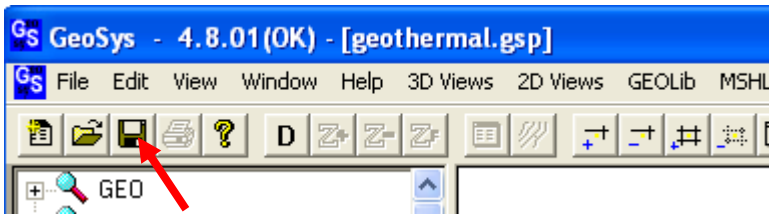


Add the mesh file into the project. Click “File” menu > “Add Member to Project …” > “MSH”. Select the geometry file “geothermal3d.msh” you created with Gina.

Imported mesh will be displayed as below.



Click “Save”.



Notice: Don't forget to save data whenever you modified something.

5 Simulation of Hydraulic process (H)

In this chapter, you will setup and simulate the fluid flow process in the reservoir. Initial condition and boundary conditions used here are presented in Figure 4 as wells as material properties in Table 1. The material properties are also used in the next chapter.

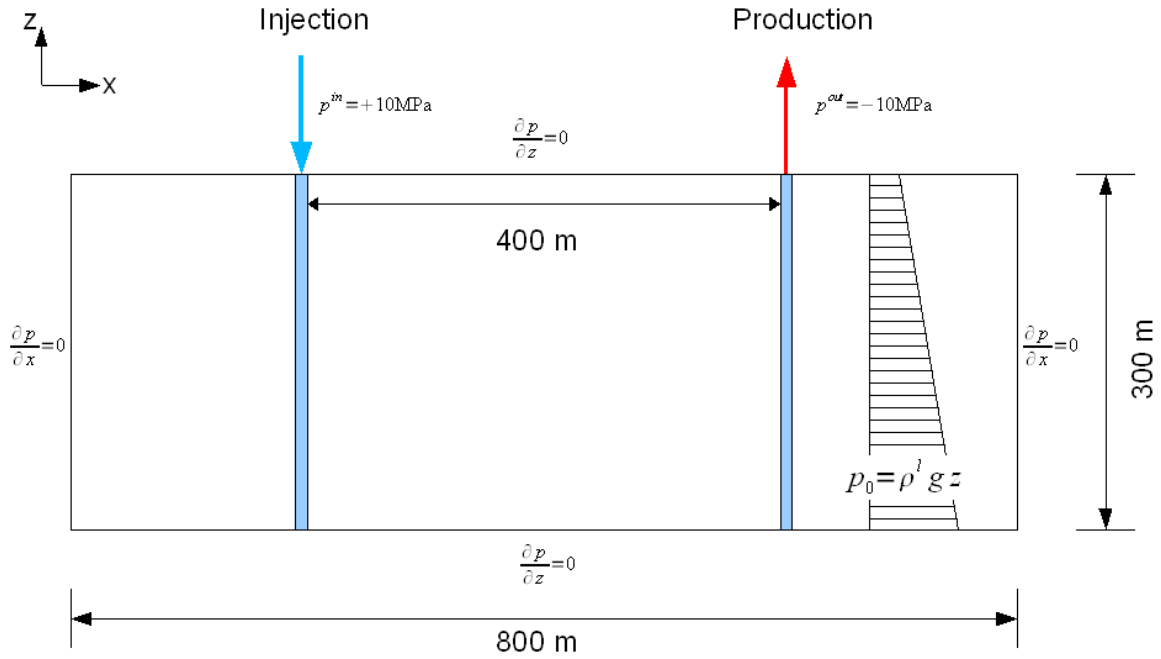


Figure 4. Initial condition and boundary condition

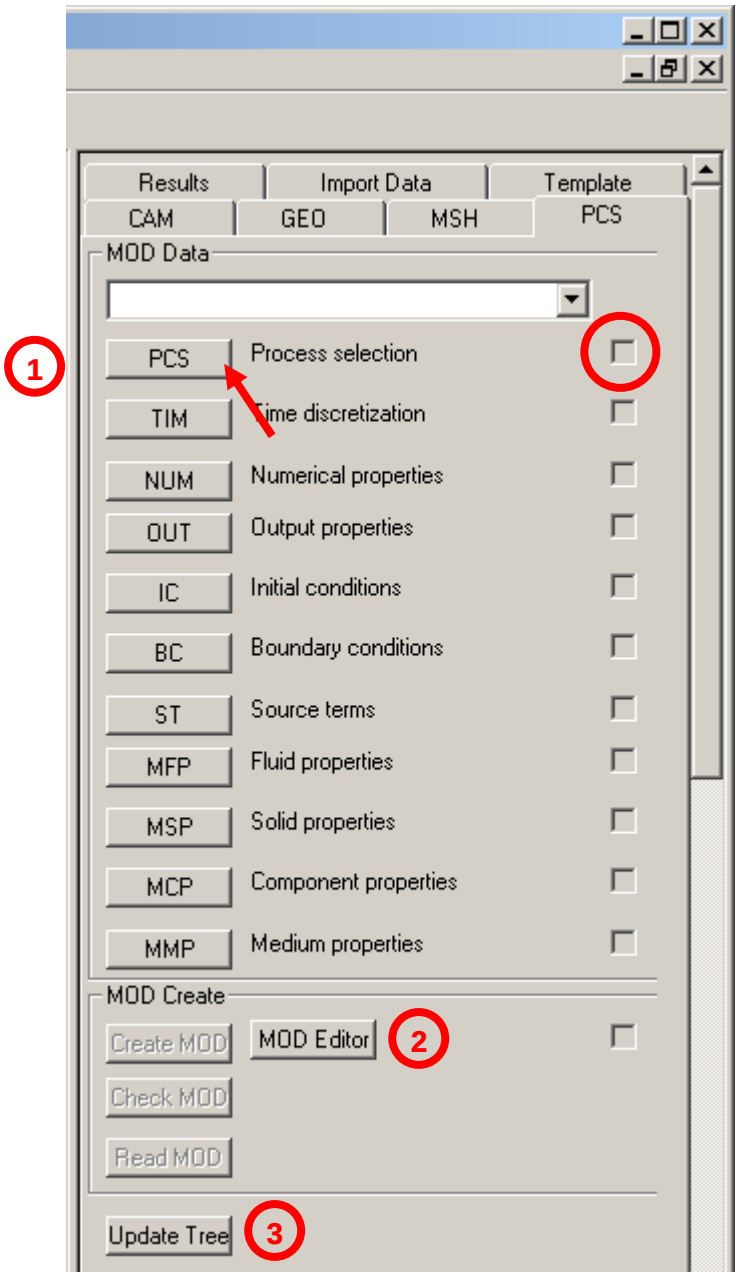
Table 1. Material properties

Porous medium properties(MMP)	
Porosity	0.5 %
Permeability	k=1.53E-15 m^2
Specific storage	1.0E-10 Pa^{-1}
Rock/Solid properties (MSP)	
Density	2850 kg/m^3
Specific heat capacity	850 J/(kg K)
Heat conductivity	3.0 W/(m K)
Young's modulus	60 GPa
Poisson ratio	0.25
Thermal expansion coefficient	1.0E-5 K^{-1}
Fluid properties (MFP)	
Density	1000 kg/m^3
Dynamic viscosity	0.001 Pa s
Specific heat capacity	4680 J/(kg K)
Heat conductivity	0.6 W/(m K)

PCS Editor

Using the PCS editor we can assign data for all PCS objects which can be subdivided into 4 types of data.

PCS	Starting point: process selection
TIM	General data for simulation control
NUM	
OUT	
IC	Data related to geometries in order to assign initial and boundary conditions as well as source/sink terms
BC	
ST	
MFP	Physico-chemical material data such as fluid, solid, chemical, and porous medium properties.
MSP	
MCP	
MMP	



Due to the character of PCS objects the according dialogs look similar (the dialog classes even are derived from a common basis class).

1

Now we go step-by-step (top down) through the PCS data dialogs, fill, and complete them. If the creation of the object data was successful we should a mark on the right hand sight.

2

The model editor (MOD Editor) is used later for combining processes, i.e. flow (H) and heat (T) to a heat transport (HT) process.

3

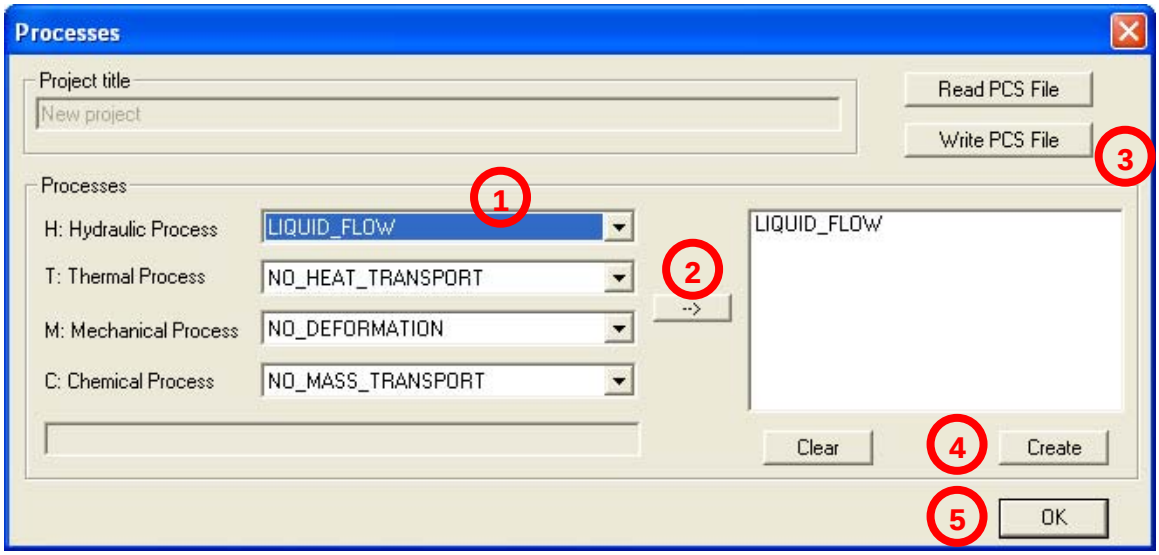
Using the Update Tree function we can refresh the Tree Window (left hand side) with the newly created data.

OGS is a (non-commercial) scientific developer software, i.e. it can crash, therefore we recommend to save the data time after time …

5.1 Creating LIQUID_FLOW process

Go to “PCS” tab on the right side and click “PCS” button.

Create LIQUID_FLOW process as follows.



Write the PCS File
Save the data.

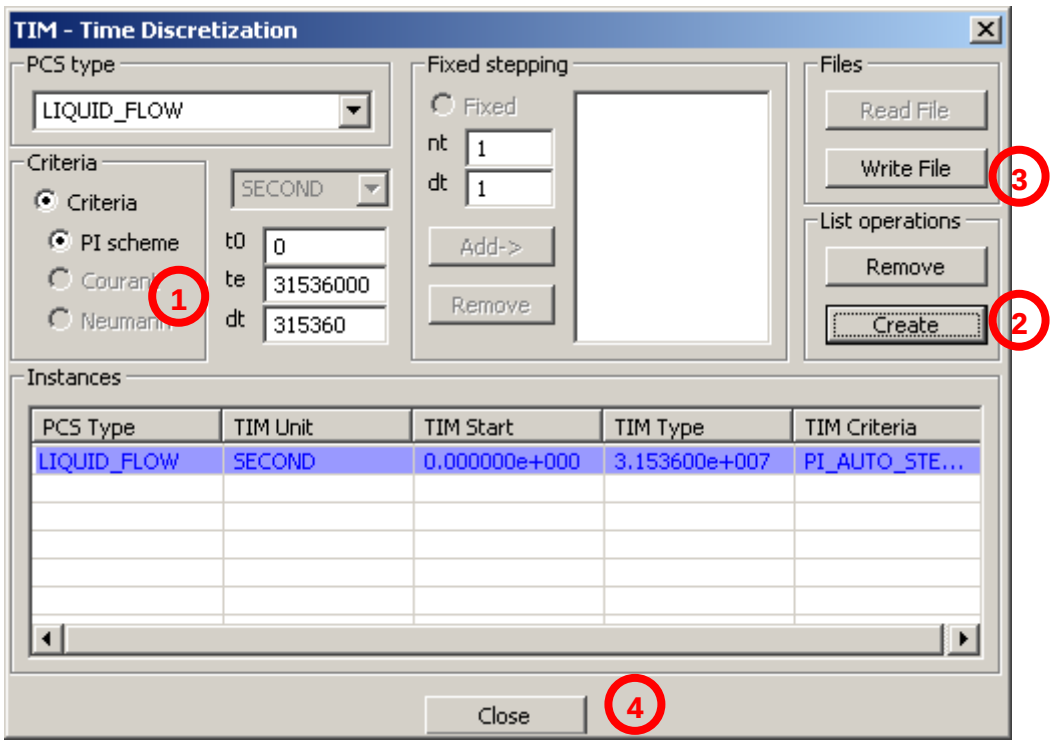
The PCS file should look like this.

```
#PROCESS
 $PCS_TYPE
  LIQUID_FLOW
 $NUM_TYPE
  FEM
 $CPL_TYPE
  PARTITIONED
 $TIM_TYPE
  TRANSIENT
#STOP
```

5.2 Time discretization

Click “TIM” button on the PCS tab.

We use an automatic (PI) time adaptation scheme which is based on energy norms for calculation optimum time steps.



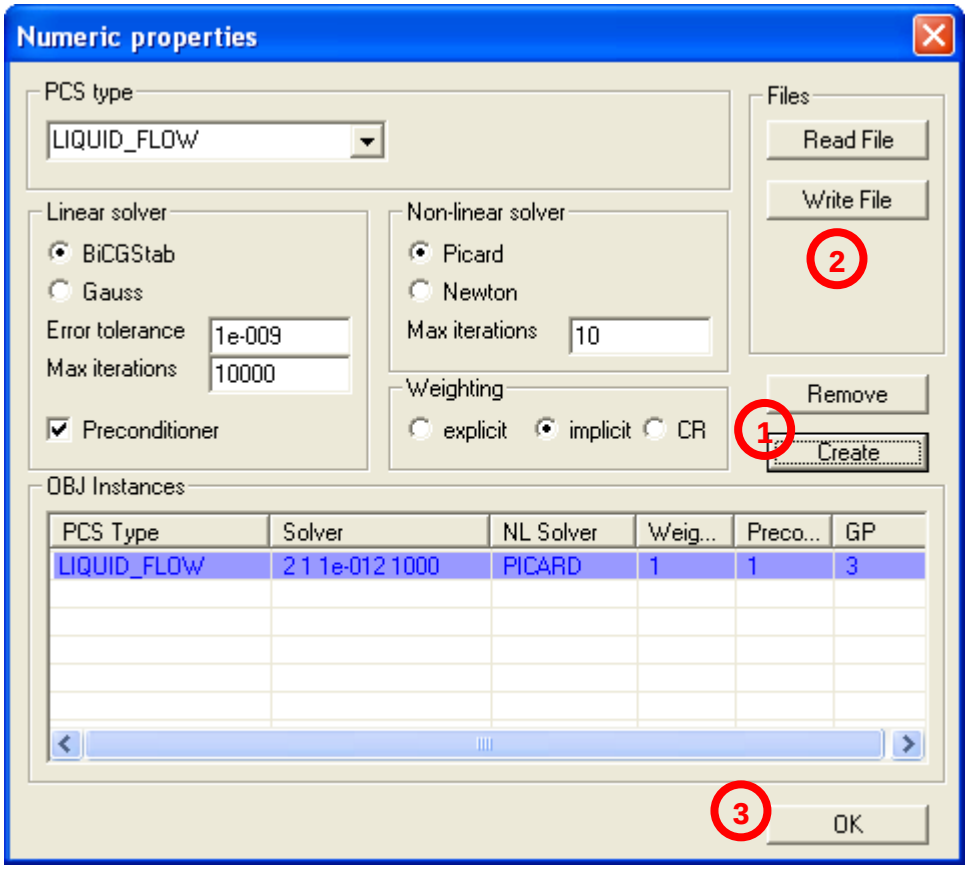
The TIM file should look like this.

```
GeoSys-TIM: ----------------------------------------------
#TIME_STEPPING
 $PCS_TYPE
  LIQUID_FLOW
 $TIME_START
  0.000000000000e+000
 $TIME_END
  3.153600000000e+007
 $TIME_CONTROL
  PI_AUTO_STEP_SIZE
   2 1.000000000000e+000 1.000000000000e-012 3.153600000000e+005
#STOP
```

5.3 Numerical properties

Click “NUM” button on the PCS tab.

We use the default settings (iterative BiCGSTAB solver)

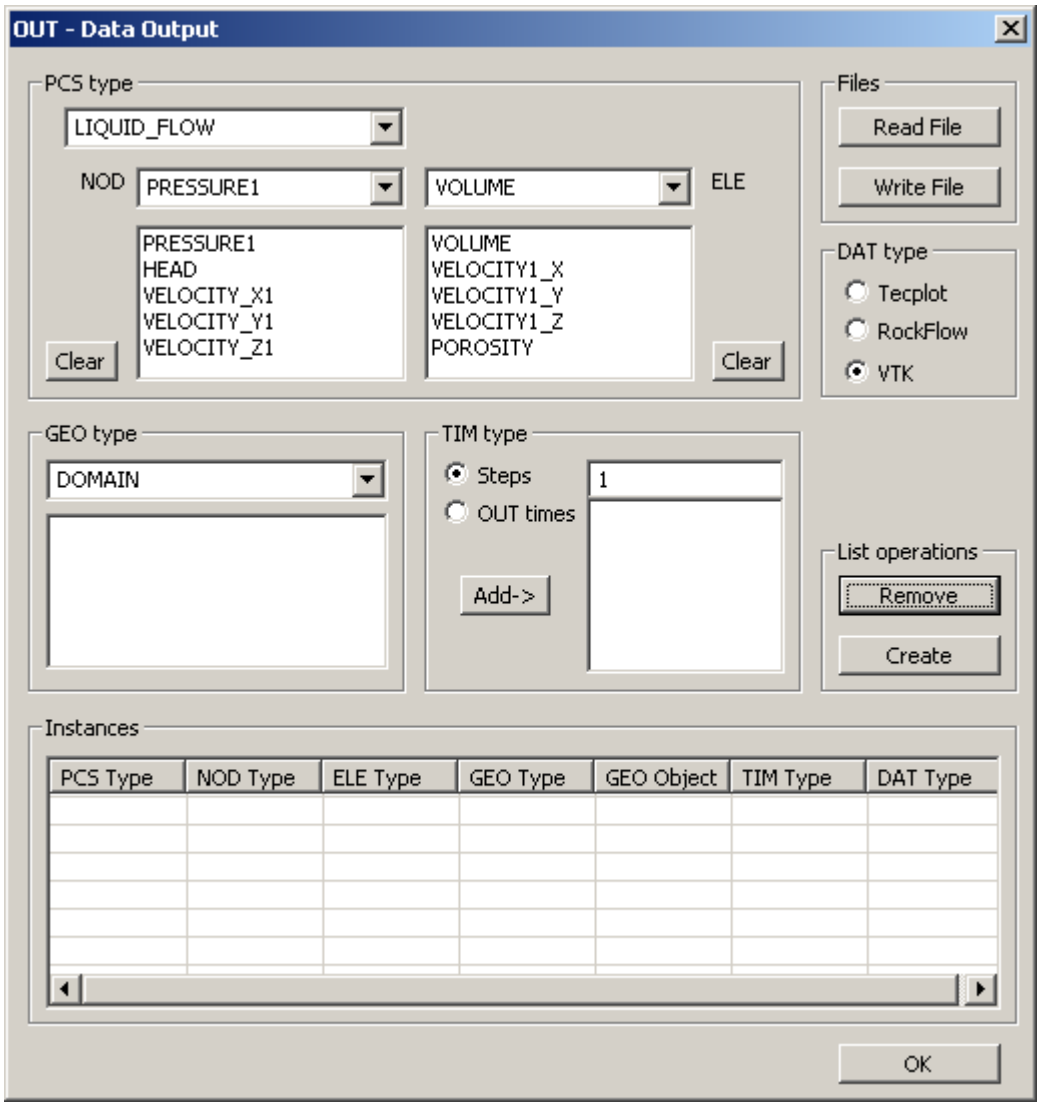


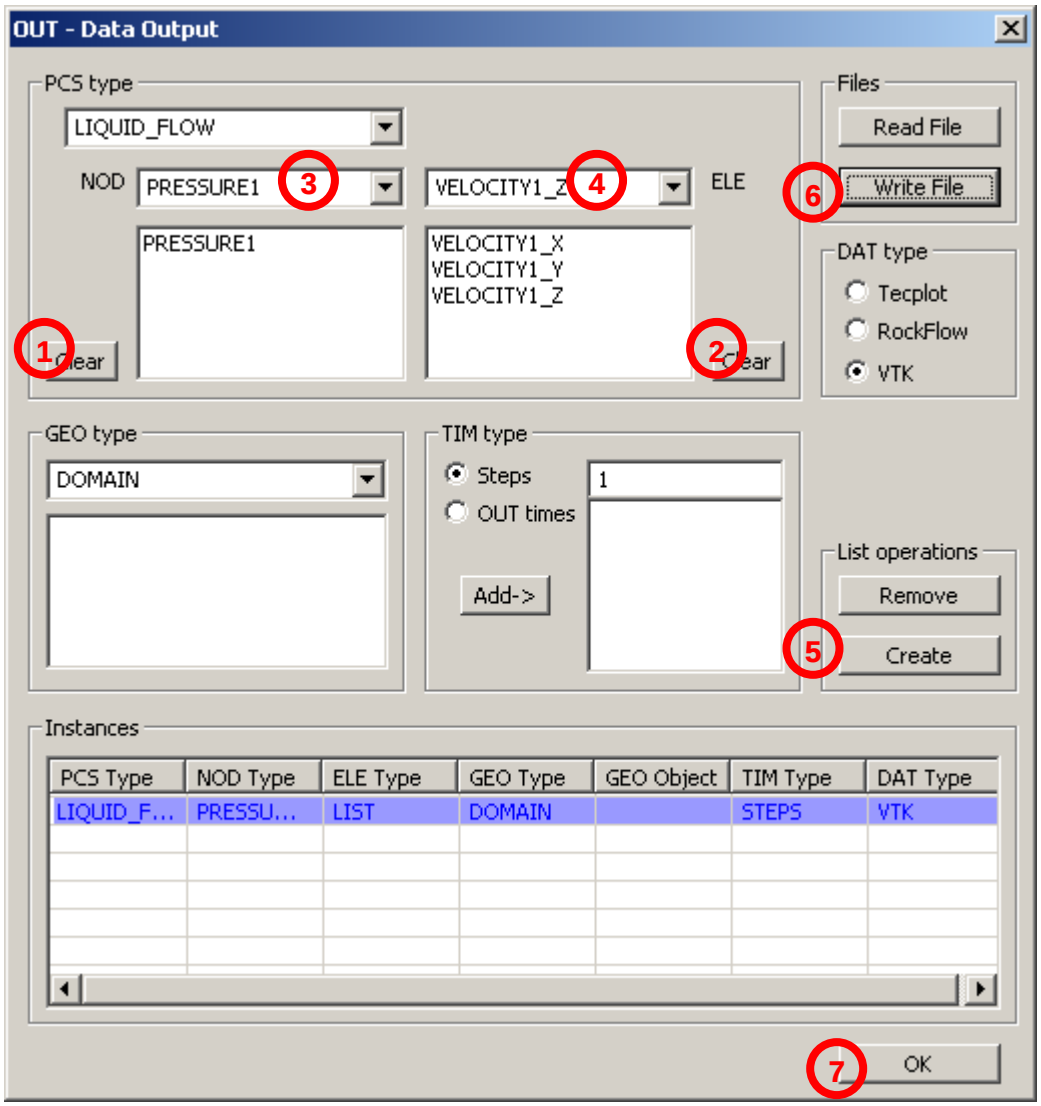
The NUM file should look like this.

```
GeoSys-NUM: Numerics ------------------------------------------
#NUMERICS
 $PCS_TYPE
  LIQUID_FLOW
 $NON_LINEAR_SOLVER
  PICARD 1.000000000000e-004 1 0.000000000000e+000
 $LINEAR_SOLVER
  2 1 1.000000000000e-012 1000 1.000000000000e+000 1 2
 $ELE_GAUSS_POINTS
  3
 $ELE_MASS_LUMPING
  0
 $ELE_UPWINDING
  0.000000000000e+000
#STOP
```

5.4 Output properties

Here you can specify which values to be outputted into result files. Click “OUT” button on the PCS tab.





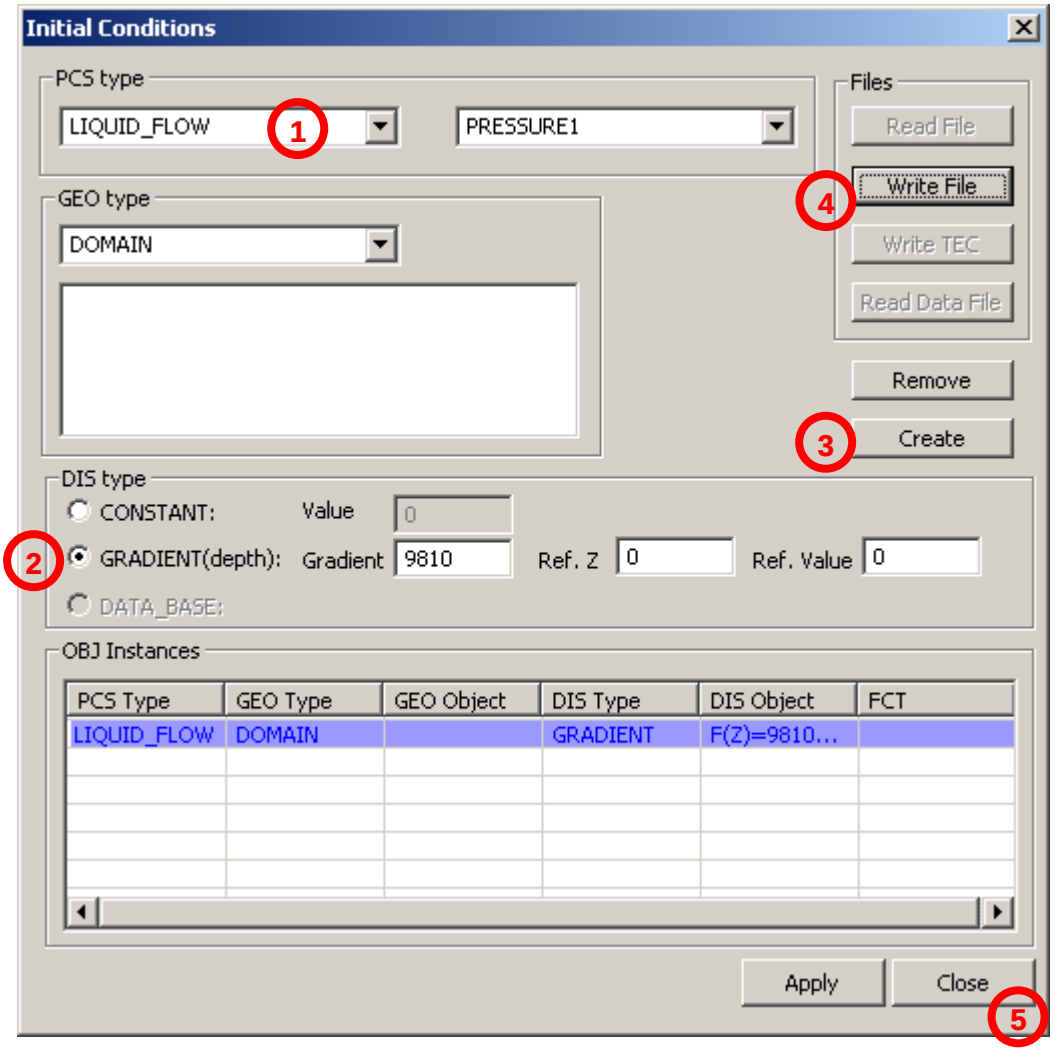
The OUT file should look like this.

```
GeoSys-OUT: Output ----------------------------------------------
#OUTPUT
 $PCS_TYPE
  LIQUID_FLOW
 $NOD_VALUES
  PRESSURE1
 $ELE_VALUES
  VELOCITY1_X
  VELOCITY1_Y
  VELOCITY1_Z
 $GEO_TYPE
  DOMAIN
 $TIM_TYPE
  STEPS 1
 $DAT_TYPE
  VTK
#STOP
```

5.5 Initial Conditions

Click “IC” button on the PCS tab.

Set linear depth-dependent hydrostatic pressure ($p_0(z) = \rho' gz$) as initial condition for the whole domain.



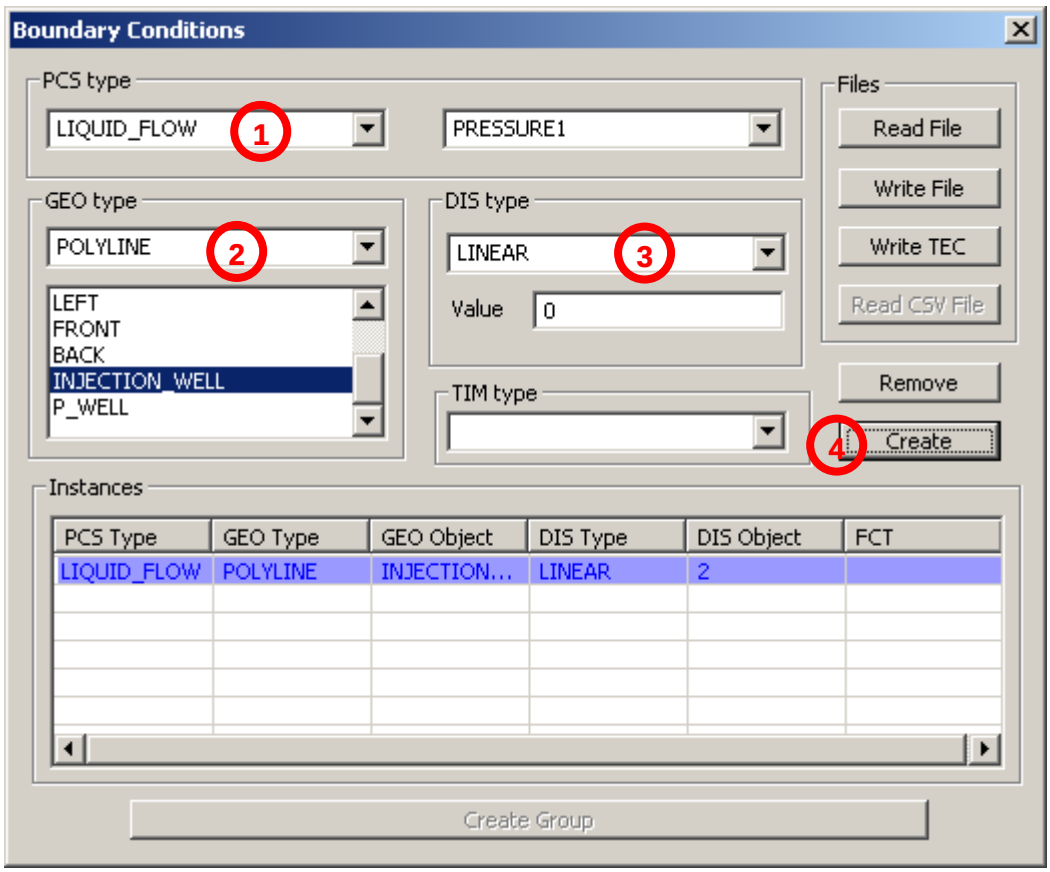
Write IC file
Save data.

```
GeoSys-IC: Initial Conditions ------------------------------------------
#INITIAL_CONDITION
 $PCS_TYPE
  LIQUID_FLOW
 $PRIMARY_VARIABLE
  PRESSURE1
 $GEO_TYPE
  DOMAIN
 $DIS_TYPE
  GRADIENT 0.000000000e+000 0.000000000e+000 9.810000000e+003
#STOP
```

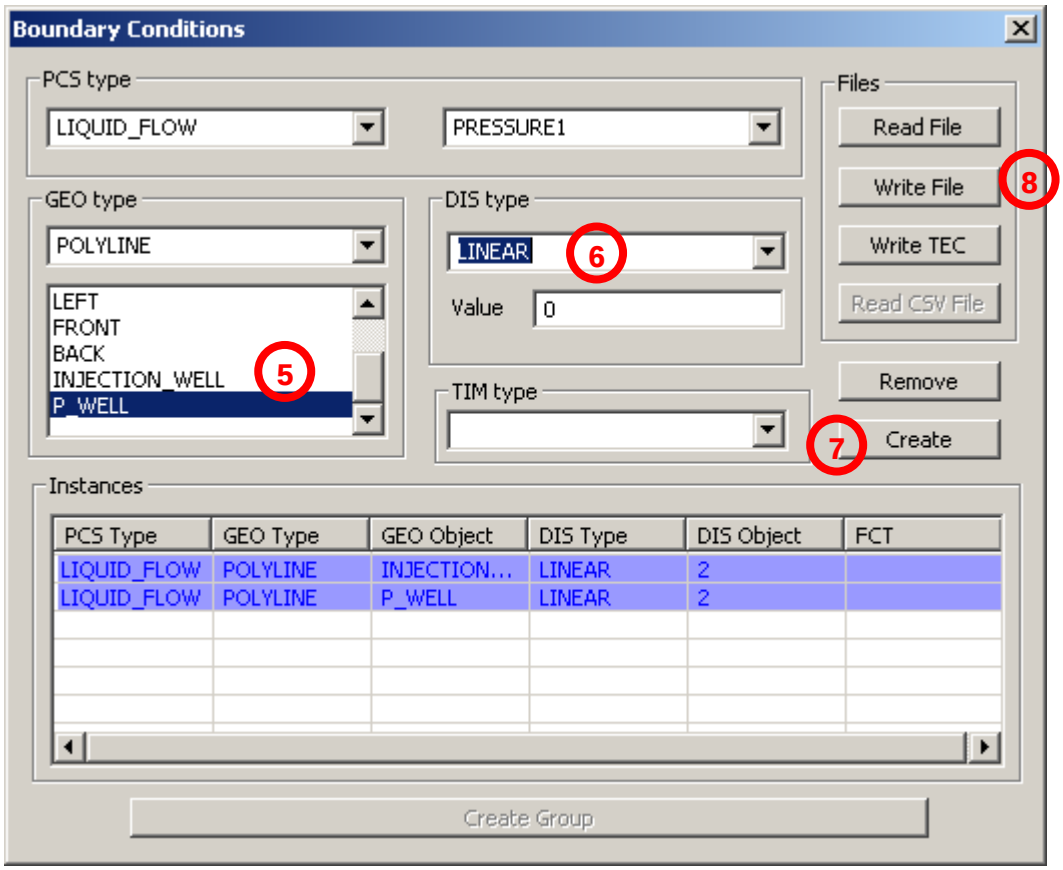
5.6 Boundary Conditions

Set constant pressure (depth-dependent) at the injection and production wells (POLYLINE: INJECTION_WELL, PRODUCTION_WELL).

Click “BC” button on the PCS tab.



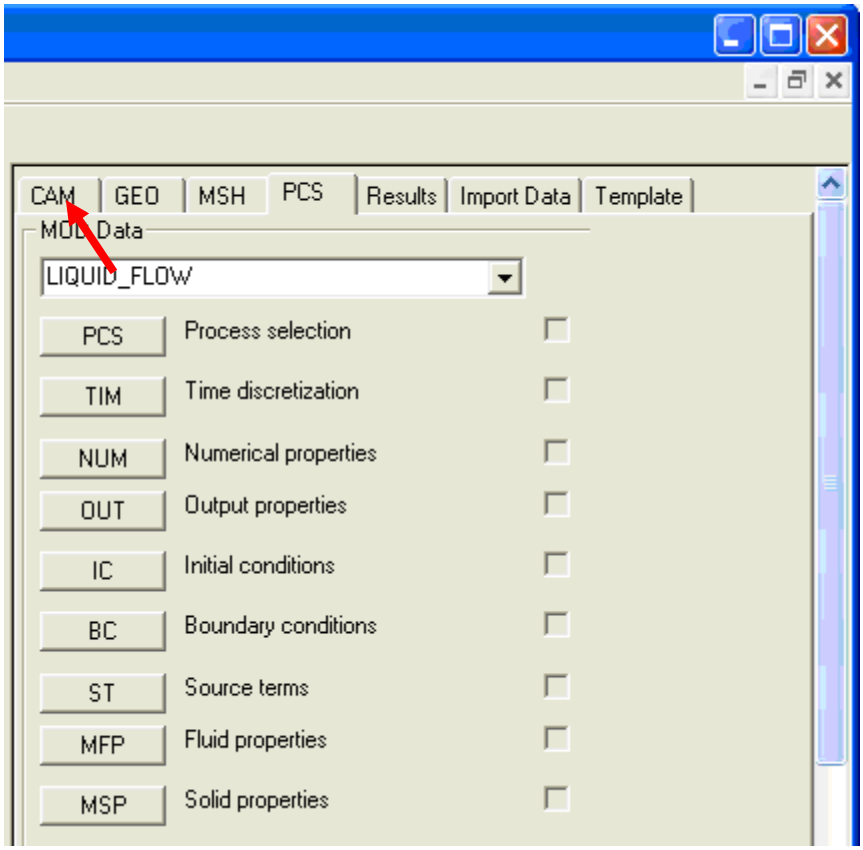

```
GeoSys-BC: Boundary Conditions ------------------------------------------
#BOUNDARY_CONDITION
 $PCS_TYPE
  LIQUID_FLOW
 $PRIMARY_VARIABLE
  PRESSURE1
 $GEO_TYPE
  POLYLINE INJECTION_WELL
 $DIS_TYPE
  LINEAR 2
  0   0.000000000000e+000
  0   0.000000000000e+000
#STOP
```



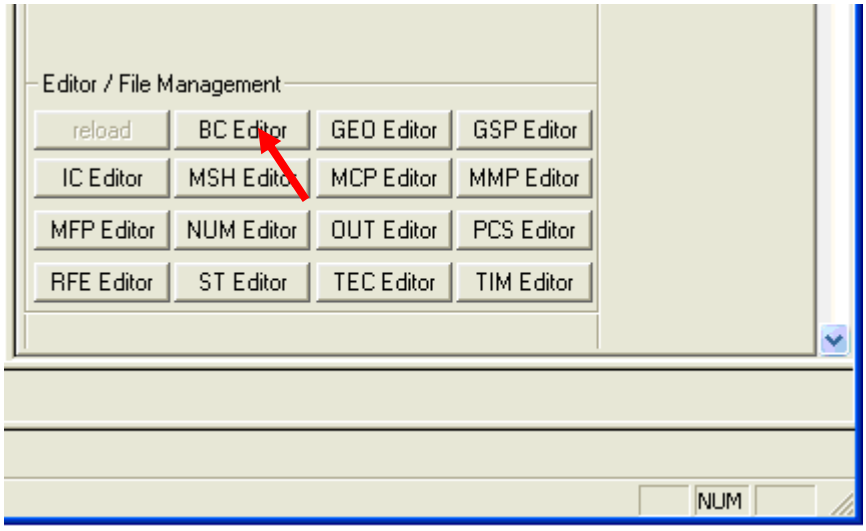
Write BC file and save data.

```
GeoSys-BC: Boundary Conditions ------------------------------------------
#BOUNDARY_CONDITION
 $PCS_TYPE
  LIQUID_FLOW
 $PRIMARY_VARIABLE
  PRESSURE1
 $GEO_TYPE
  POLYLINE INJECTION_WELL
 $DIS_TYPE
  LINEAR 2
  0   0.000000000000e+000
  0   0.000000000000e+000
#BOUNDARY_CONDITION
 $PCS_TYPE
  LIQUID_FLOW
 $PRIMARY_VARIABLE
  PRESSURE1
 $GEO_TYPE
  POLYLINE P_WELL
 $DIS_TYPE
  LINEAR 2
  0   0.000000000000e+000
  0   0.000000000000e+000
#STOP
```

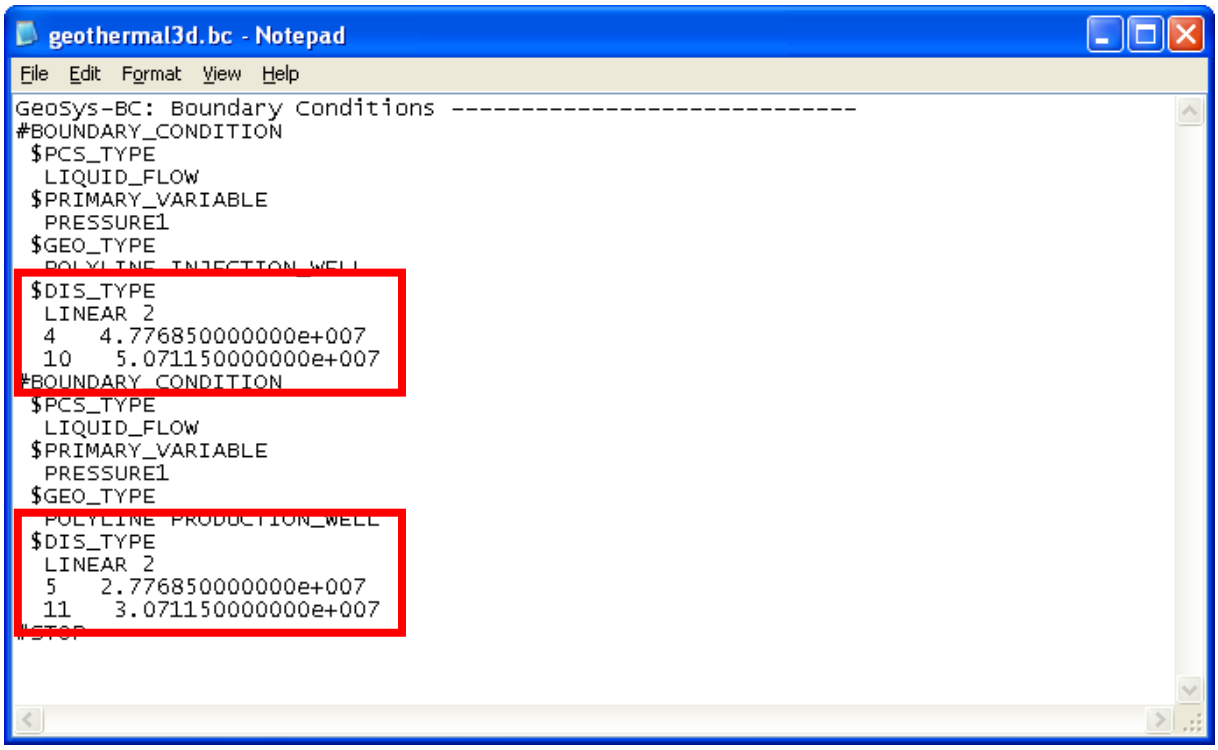
Go to the CAM tab.



Click “BC Editor” button. (You may need to scroll down in the CAM tab to find the button.)



Modify the below sections in the file.



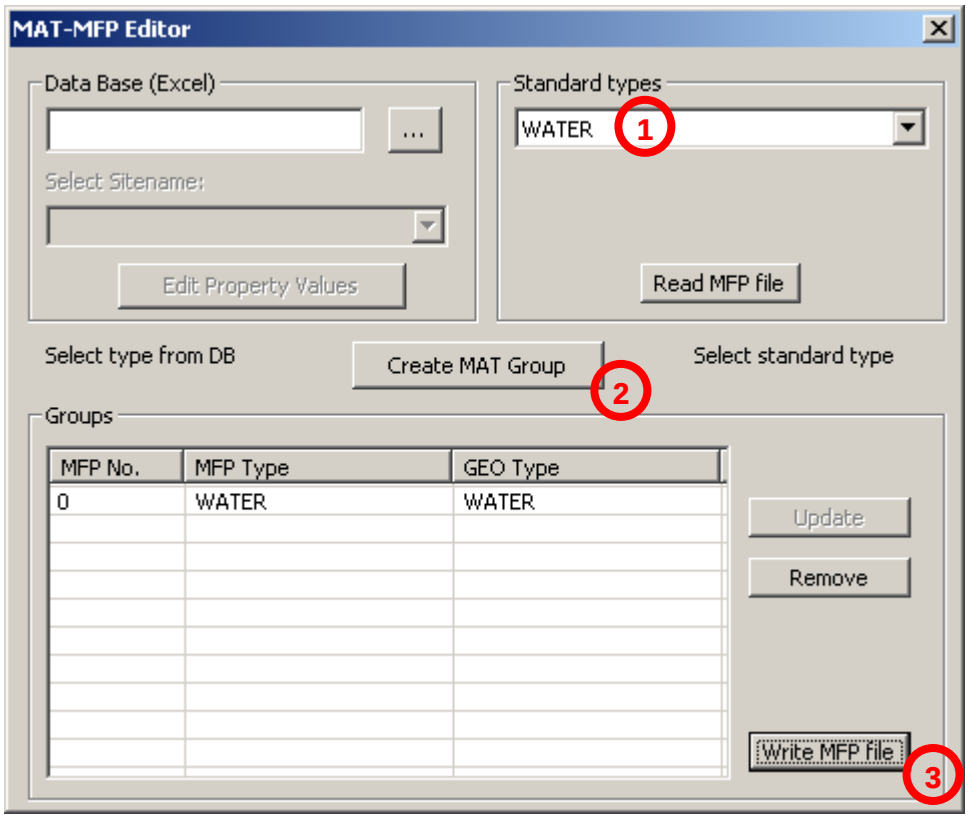

```
$DIS_TYPE
  LINEAR 2
  4   4.776850000000e+007
  10   5.071150000000e+007
```

```
$DIS_TYPE
  LINEAR 2
  5   2.776850000000e+007
  11   3.071150000000e+007
```

Save and close the file.
Notice: If you are using the old version of GeoSys, you need to restart GeoSys without saving the project and reload the project. Otherwise, GeoSys will initialize the BC file when saving the project.

5.7 Material properties (MFP / MSP / MMP)

We consider single phase flow of water through the geothermal reservoir. We can create the standard fluid properties for water as follows.



MFP file should look like this

```
GeoSys-MFP: Material Fluid Properties ------------
 #FLUID_PROPERTIES
  $FLUID_TYPE
   WATER
  $DAT_TYPE
   WATER
  $DENSITY
   1 1.000000000000e+003
  $VISCOSITY
   1 1.000000000000e-003
  $SPECIFIC_HEAT_CAPACITY
   1 4.680000000000e+003
  $SPECIFIC_HEAT_CONDUCTIVITY
   1 6.000000000000e-001
 #STOP
```

The specified fluid properties are density, viscosity, heat capacity and heat conductivity.

Properties of the solid phases and the porous medium are material-dependent; they can be prepared in EXCEL files and imported into GeoSys.

The first figure (msp.xls) shows the solid phase properties for crystalline rock. The left column (A) contains the keywords for the material properties, the next columns depicts the values for the material group (CRYSTALLINE) listed in the upper row.

Microsoft Excel - msp.xls

File Edit View Insert Format Tools Data Window

E12

	A	B
1		CRYSTALLINE
2	DENSITY	2.85E+03
3	YOUNGS_MODULUS	6.40E+10
4	POISSON_RATIO	2.50E-01
5	THERMAL_EXPANSION	1.00E-04
6	HEAT_CAPACITY	8.50E+02
7	HEAT_CONDUCTIVITY	2.79E+00
8		

The same is valid for the properties of the porous media (mmp.xls). Several material groups can be defined in the EXCEL table, which can be selected later on.

Microsoft Excel - mmp.xls

File Edit View Insert Format Tools Data Window Help Adobe PDF

G32 3

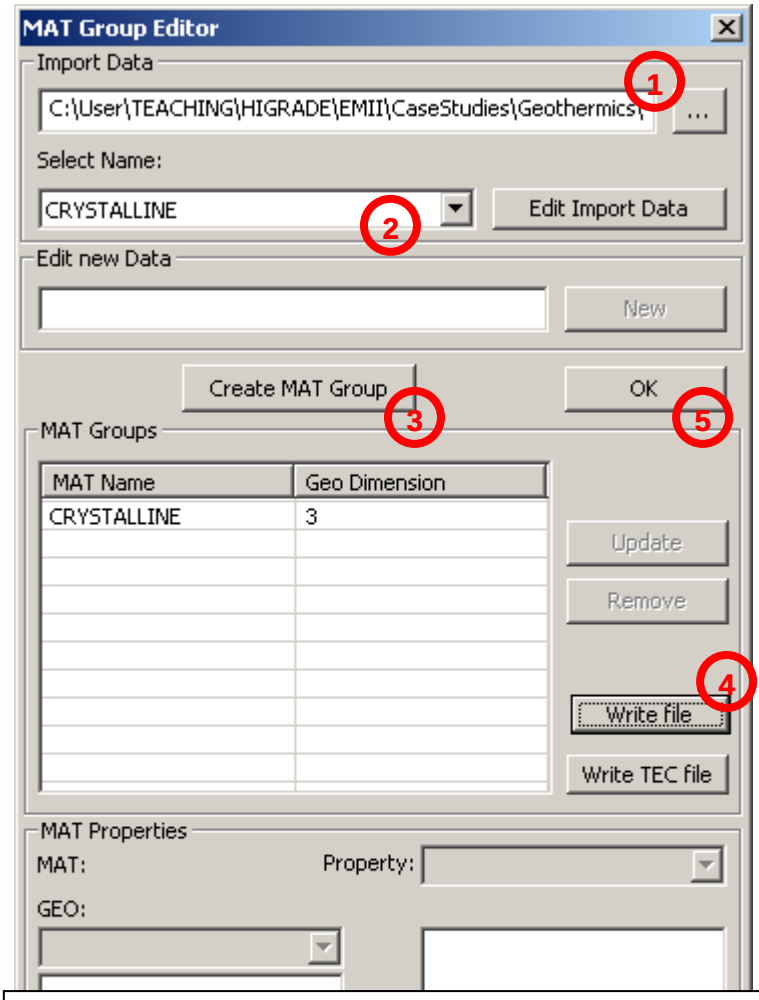
	A	B	C	D	E	F	G	H
1		SURFACE	SOIL1	SOIL2	SOIL3	AQUIFER1	AQUIFER2	CRYSTALLINE
2	POROSITY		0,3946	0,4387	0,4764	0,2	0,125	0,00405
3	TORTUOSITY							1
4	MOBILE_IMOBILE_MODEL							
5	LITHOLOGY_GRAIN_CLASS							
6	FLOWLINEARITY							
7	SORPTION_MODEL							
8	STORAGE							
9	CONDUCTIVITY							
10	PERMEABILITY_TENSOR		3,13E-12	2,47E-12	2,20E-12	1,00E-05	1,25E-05	1,53E-15
11	PERMEABILITY_FUNCTION							
12	PERMEABILITY_FUNCTION_DEFORMATION							
13	PERMEABILITY_FUNCTION_PRESSURE							
14	PERMEABILITY_SATURATION		4	4	4			
15	PERMEABILITY_FUNCTION_STRESS							
16	PERMEABILITY_FUNCTION_VELOCITY							
17	PERMEABILITY_FUNCTION_POROSITY							
18	CAPILLARY_PRESSURE		4	4	4			
19	MASSDISPERSION_LONG							
20	MASSDISPERSION_TRANS							
21	HEATDISPERSION_LONG							
22	HEATDISPERSION_TRANS							
23	ELECTRIC_CONDUCTIVITY							
24	UNCONFINED_FLOW_GROUP							
25	FLUID_EXCHANGE_WITH_OTHER_CONTINUA							
26	SATURATION_RESIDUAL		0,05068	0,05699	0,05248			
27	VAN_GENUCHTEN_ALPHA		7,00E-04	5,60E-04	4,43E-04			
28	VAN_GENUCHTEN_BETA		3,4265	4,1371	4,3565			
29	DARCY_WEISBACH_COEFFICIENT	0,91						
30	CHEZY_COEFFICIENT	9,2818						
31	MANNING_COEFFICIENT	3,40E-02						
32	DIMENSION	2	1	1	1	3	3	3

Tabelle1 Tabelle2 Tabelle3

Ready

Porous medium properties (MMP)

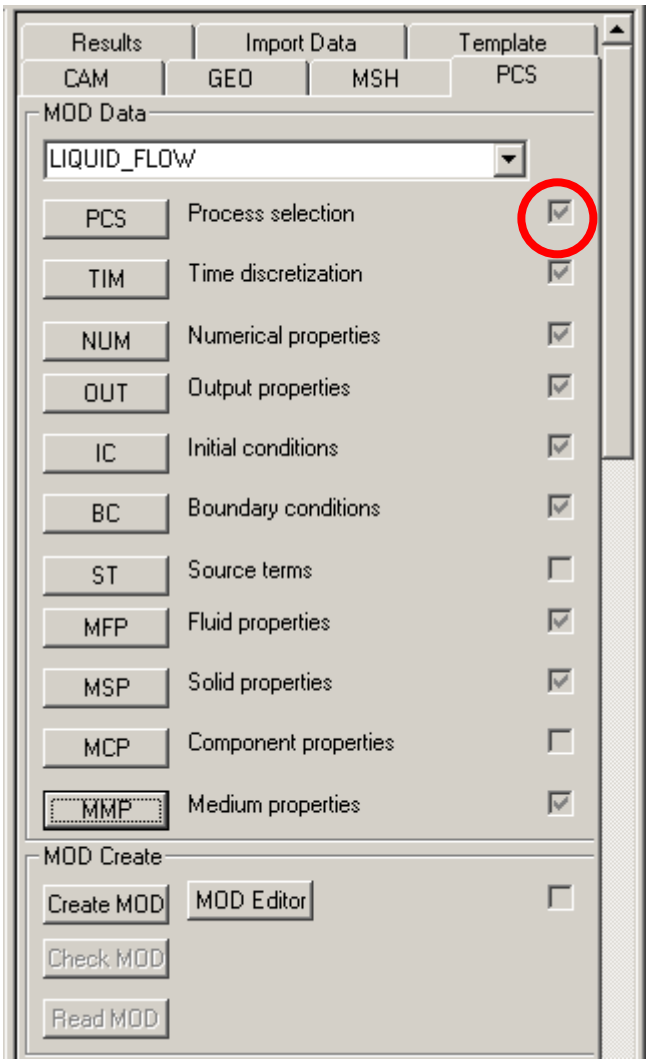
Click the MMP button on the PCS tab.
Repeat exactly the same procedure for creating the MMP group as for the MSP data.




```
GeoSys-MMP: Material Medium Properties -----------------------------------
#MEDIUM_PROPERTIES
 $NAME
  CRYSTALLINE
 $GEOMETRY_DIMENSION
  3
 $GEOMETRY_AREA
  1.000000000000e+000
 $POROSITY
  1 5.000000000000e-003
 $TORTUOSITY
  1 1.000000000000e+000
 $PERMEABILITY_TENSOR
  ISOTROPIC 1.530000000000e-015
#STOP
```

Now we have prepared all required data to run the flow simulation. Before we do so we make several checks.

(1) Have all OGS object data for the selected process been created?
For the LIQUID_FLOW process: PCS, TIM, NUM, OUT, IC, BC, MFP, and MMP data are required. There should be a mark on the right column.



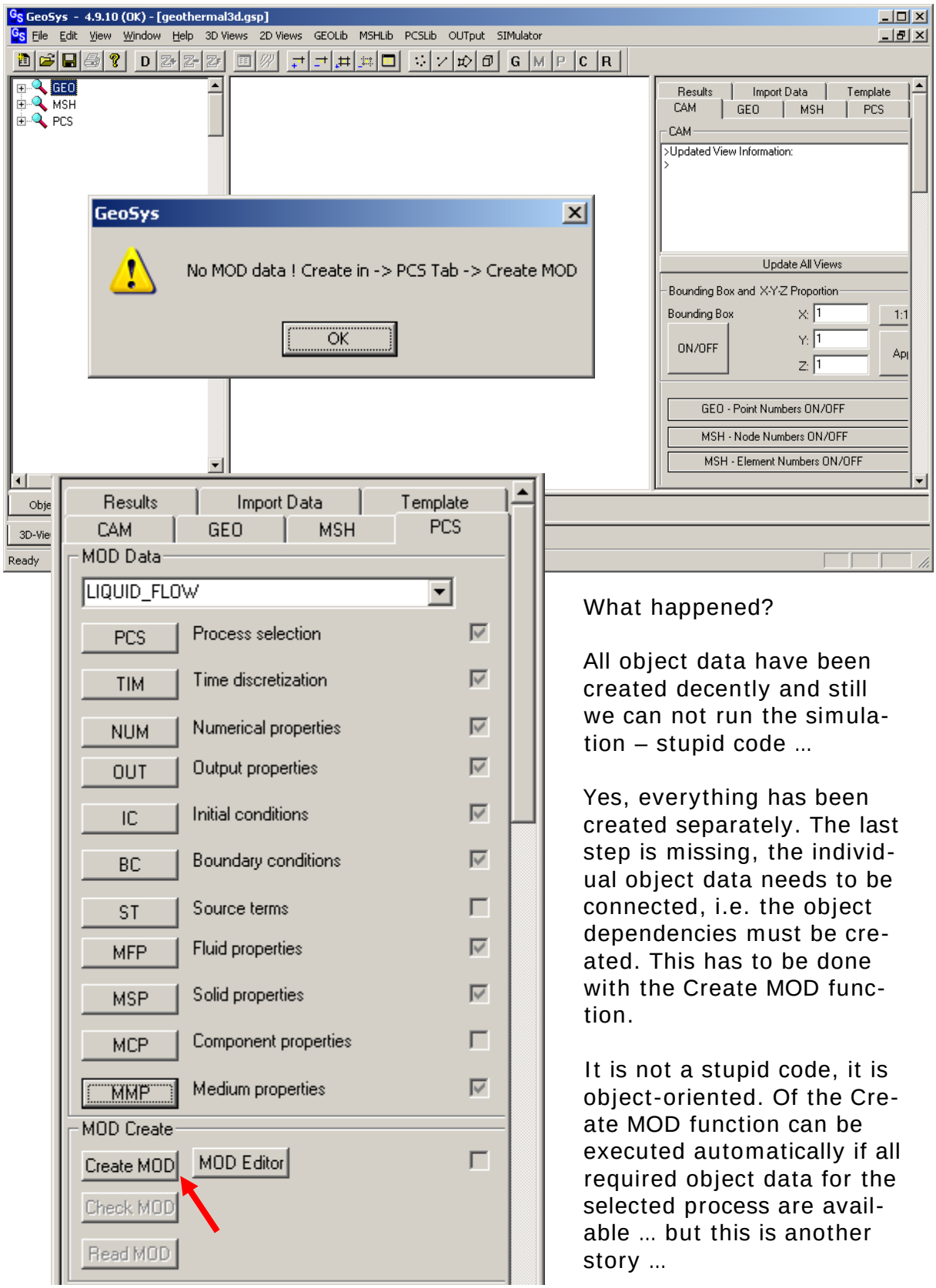
(2) Is the OGS project (GSP) file complete ?

```
#PROJECT_MEMBER
geothermal3d.gli
geothermal3d.msh
geothermal3d.pcs
geothermal3d.tim
geothermal3d.num
geothermal3d.out
geothermal3d.ic
geothermal3d.bc
geothermal3d.mfp
geothermal3d.msp
geothermal3d.mmp
#STOP
```

The GSP file should contain all object files.

(3) Creating MOD data

If you want to run the simulation now – you will receive the following Error message: no MOD data !



What happened?

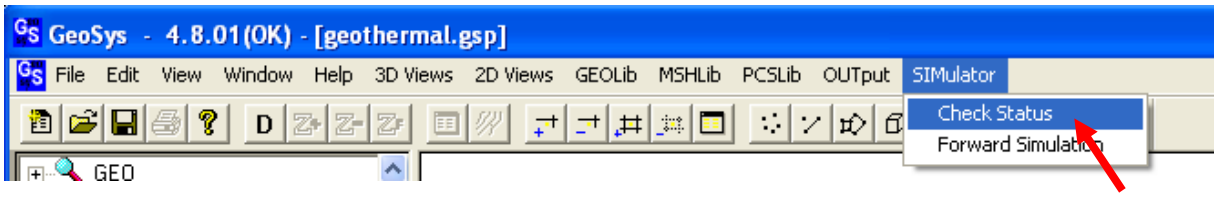
All object data have been created decently and still we can not run the simulation – stupid code …

Yes, everything has been created separately. The last step is missing, the individual object data needs to be connected, i.e. the object dependencies must be created. This has to be done with the Create MOD function.

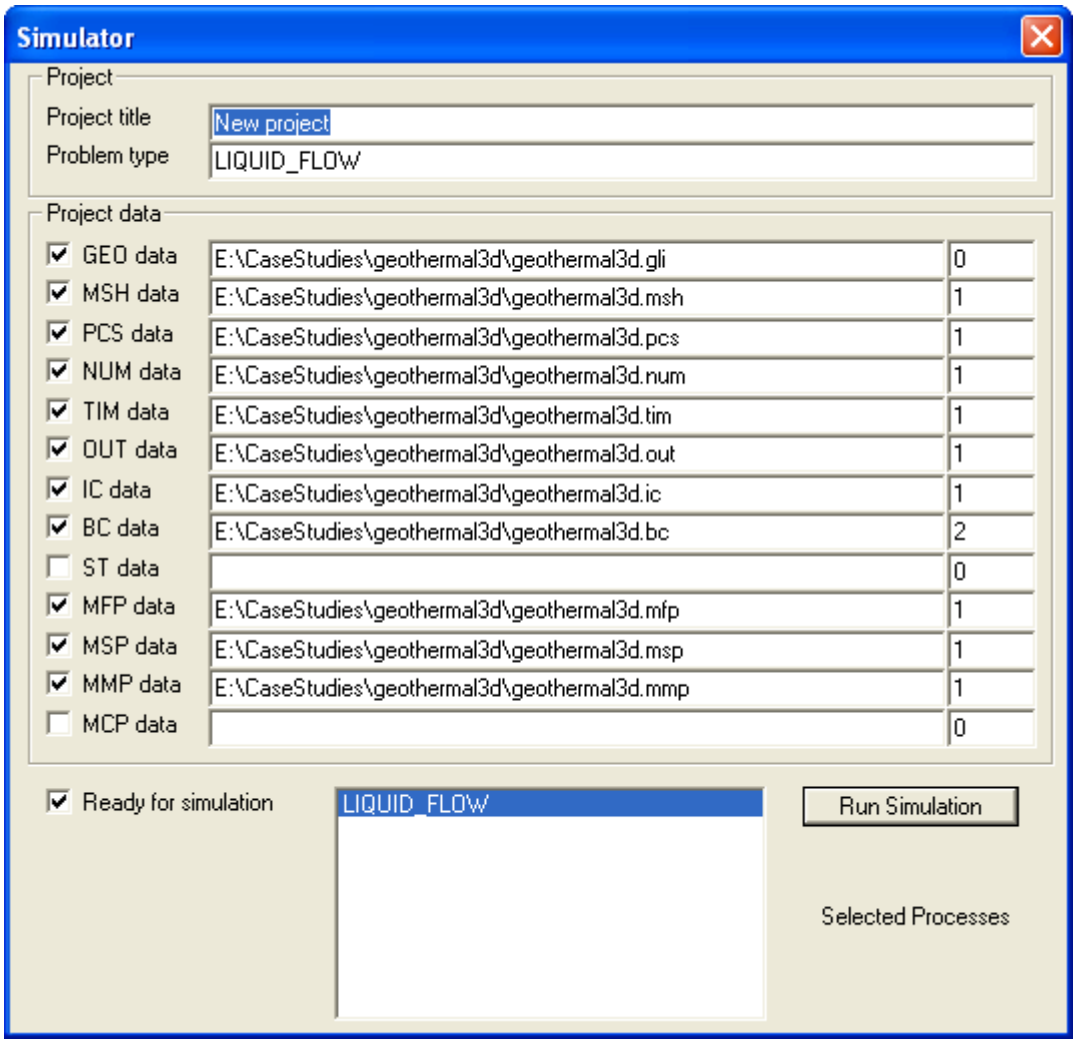
It is not a stupid code, it is object-oriented. Of the Create MOD function can be executed automatically if all required object data for the selected process are available … but this is another story …

5.8 Run simulation

Click “SIMulator” menu -> “Check Status”.



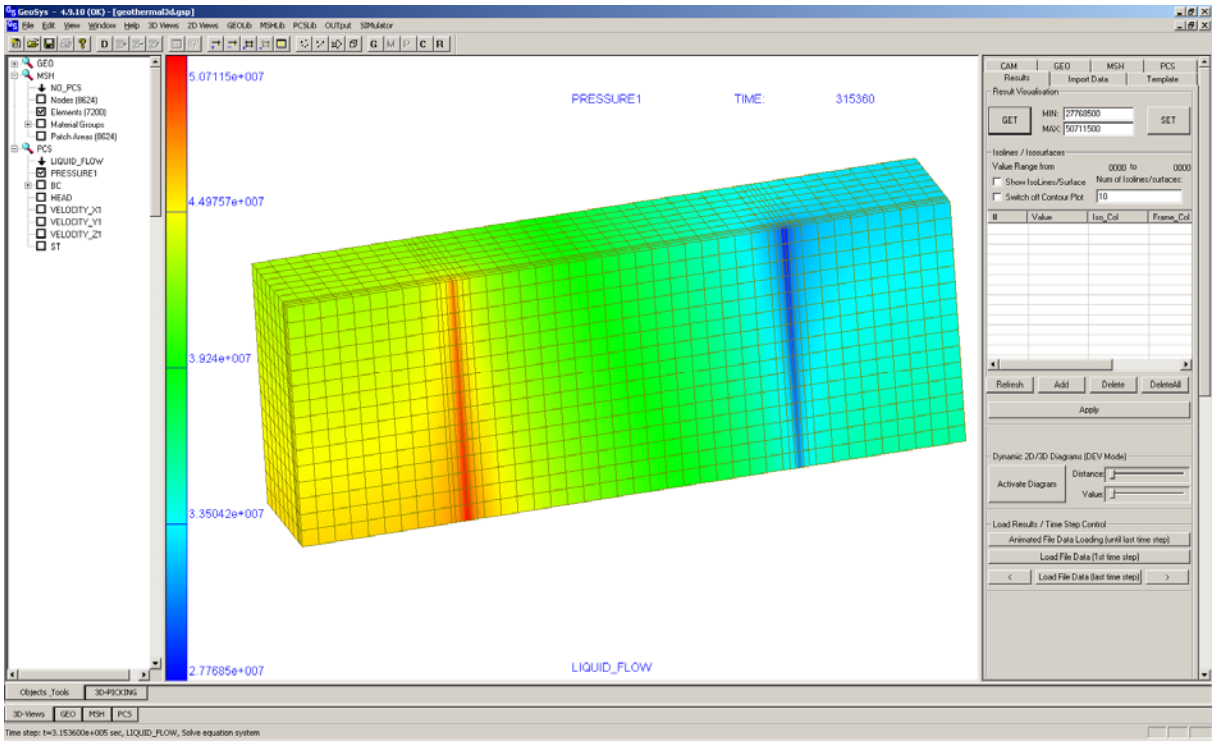
Check whether all necessary files have been prepared.



Close the dialog.

Now you are ready to run a simulation. Click “SIMulator” menu -> “Forward Simulation”. The simulation may take a few minutes.

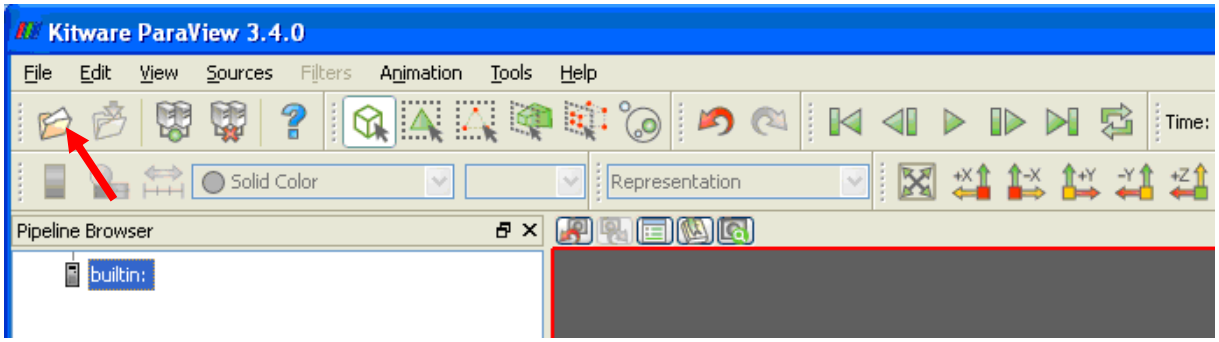
After finishing the simulation, results (pressure field) will be displayed as below.



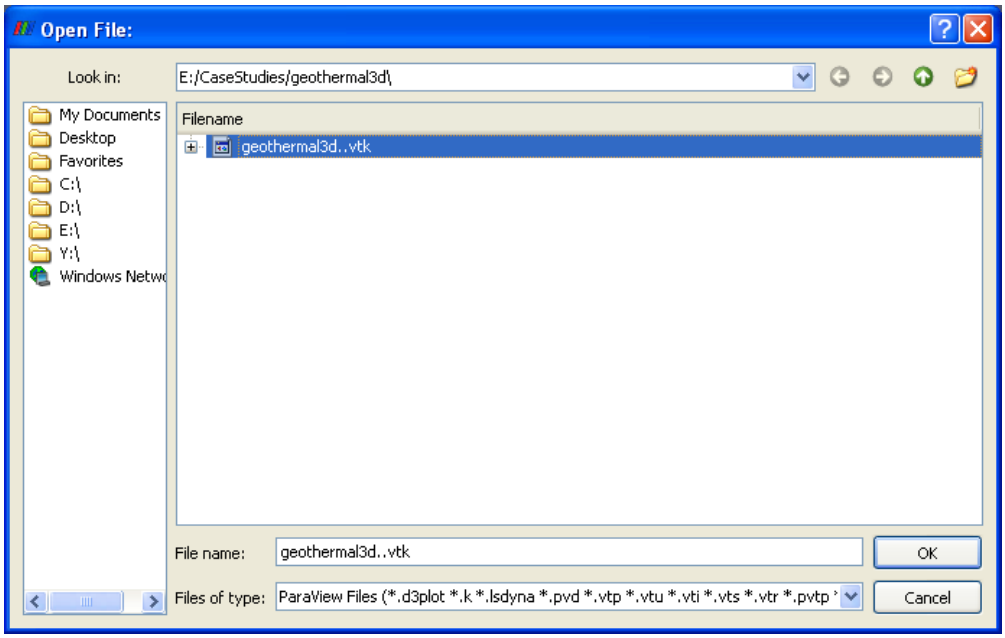
5.9 Visualization of simulation result with ParaView

You can also show simulation results using ParaView. Although GeoSys GUI is able to show simulation results, its functions are still limited. For advanced visualization, ParaView is one of alternatives.

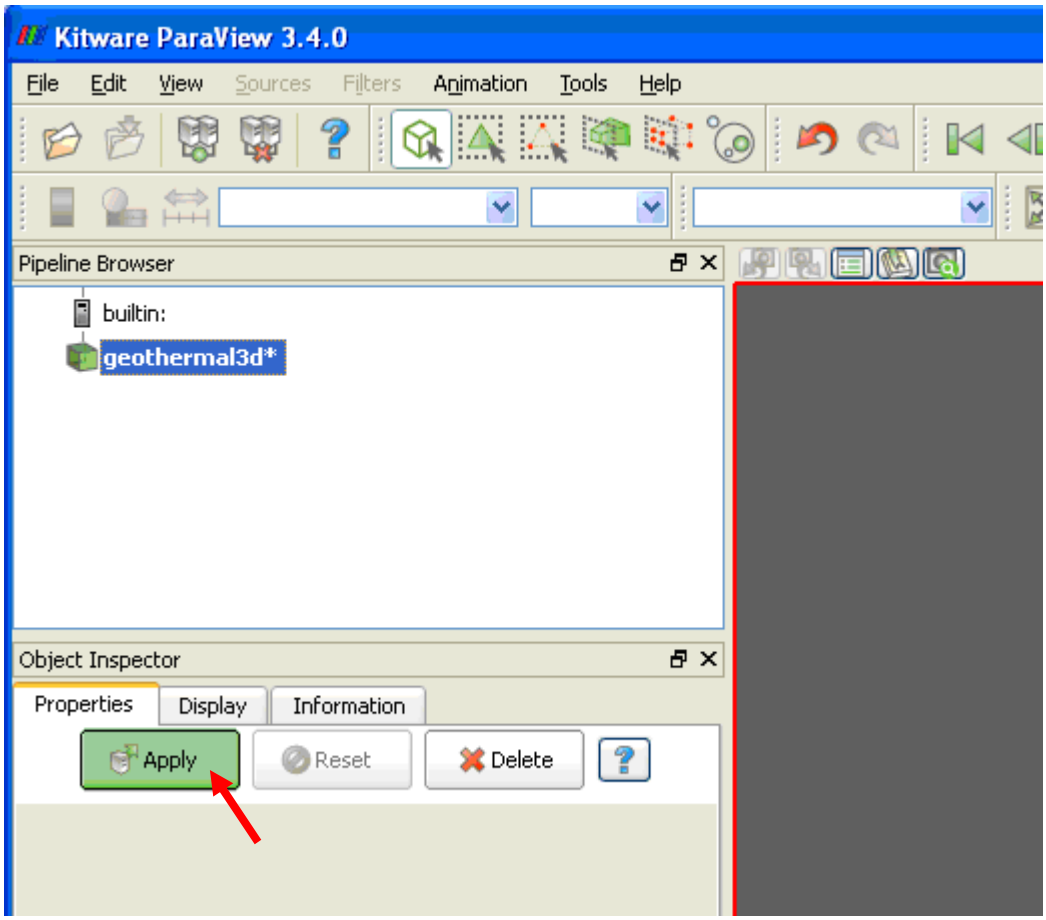
Start ParaView from Start menu and load result files (*.vtk).
Click the File Open button or click “File” menu > “open ”.

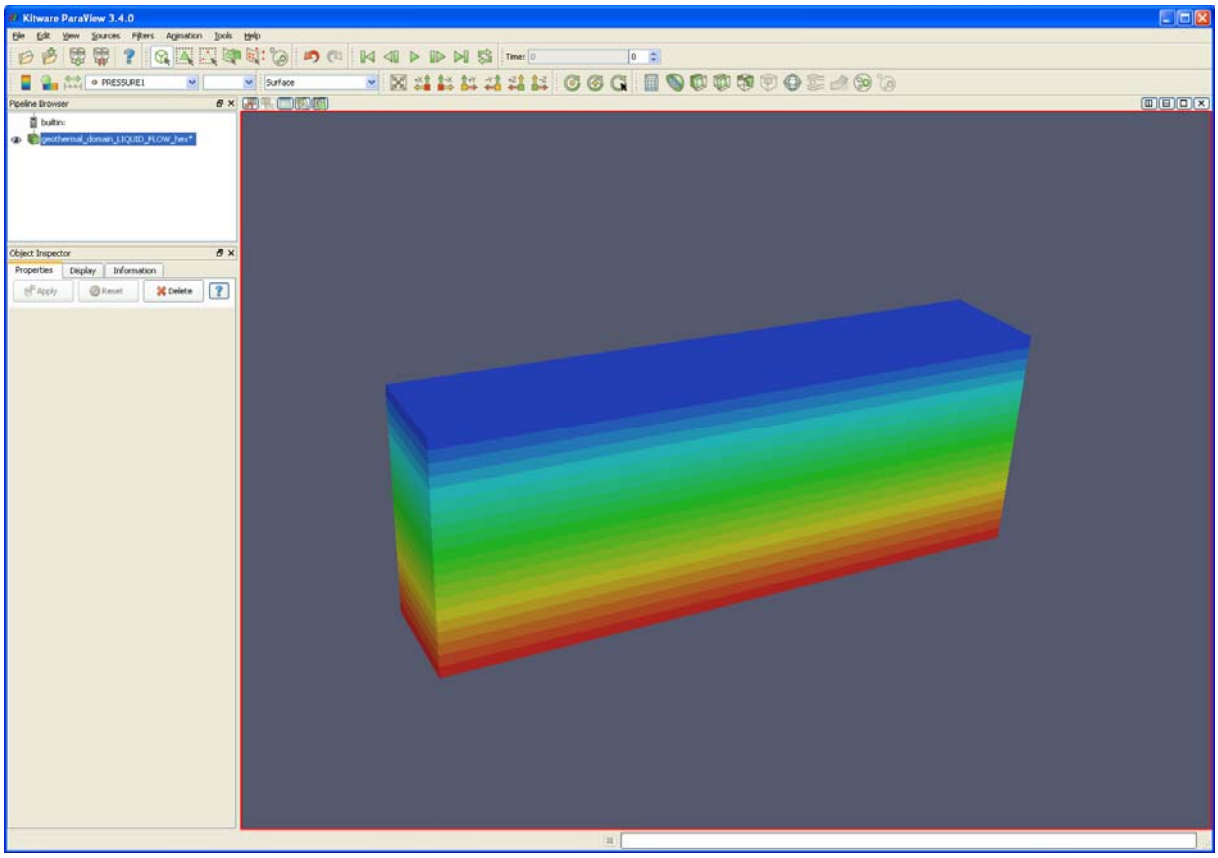


Browse the directory where result files exist. Select “geothermal3d..vtk”.

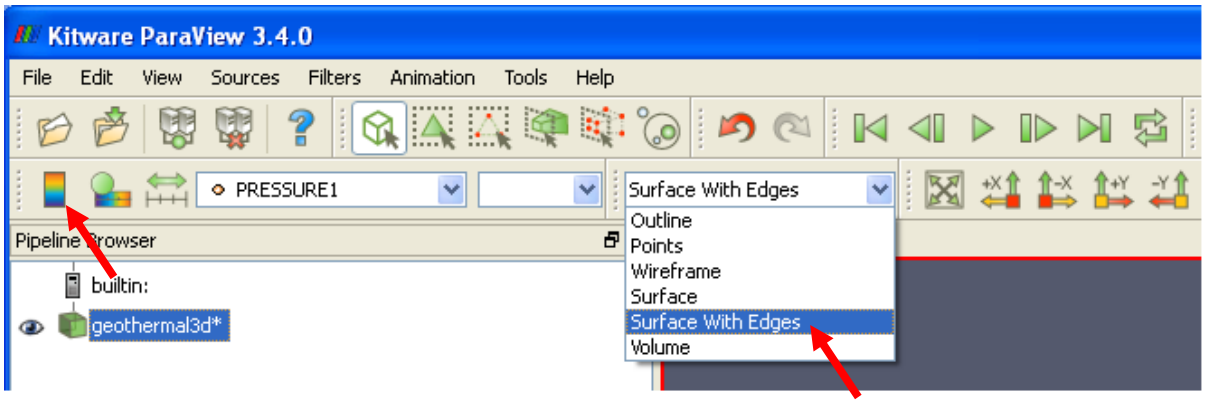


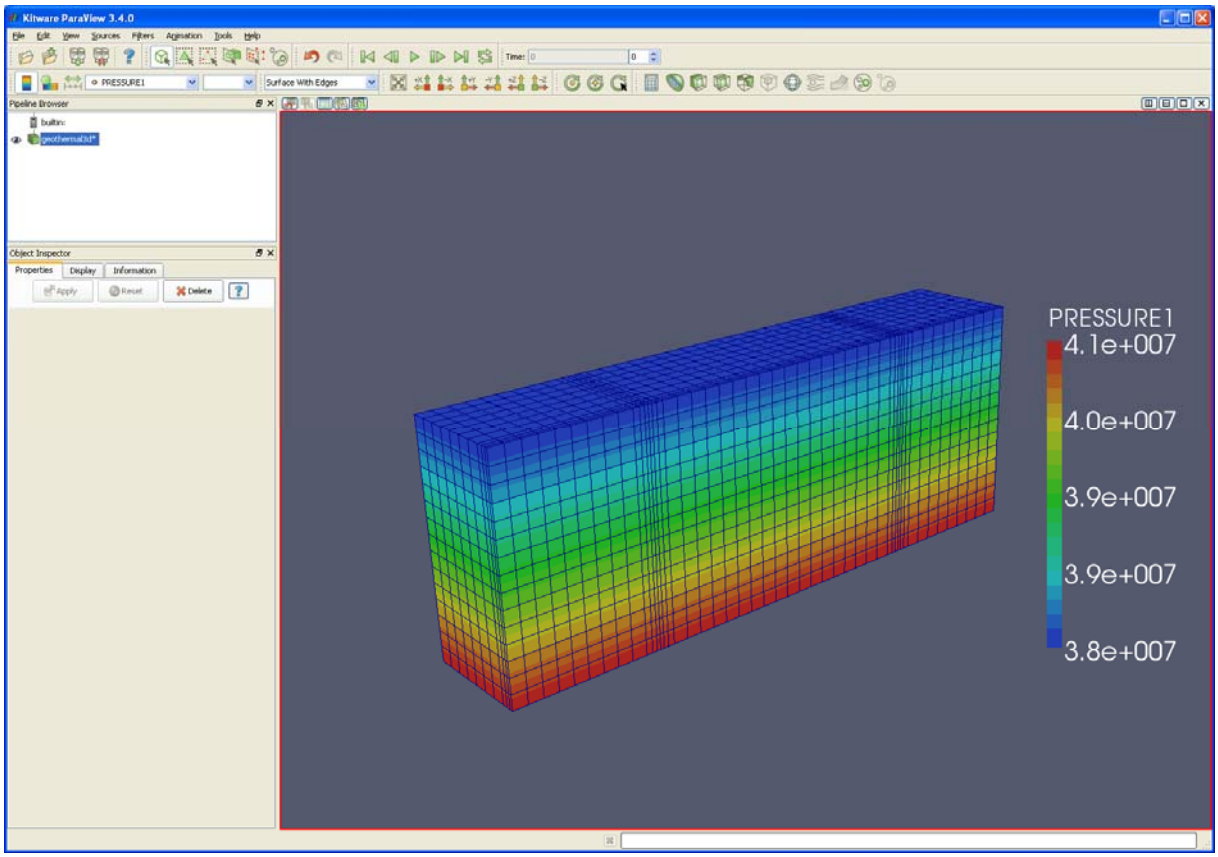
Click “Apply” button on the Object Inspector view to show results.



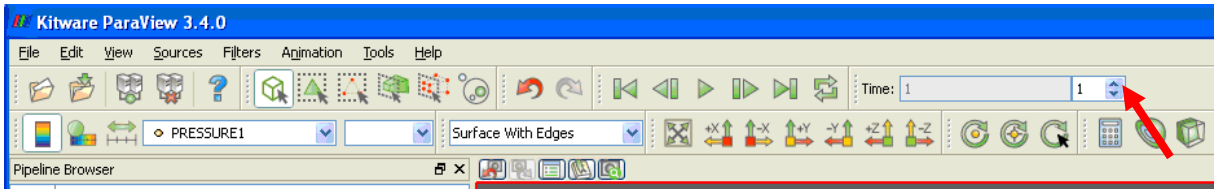


You can display mesh edges and legends.

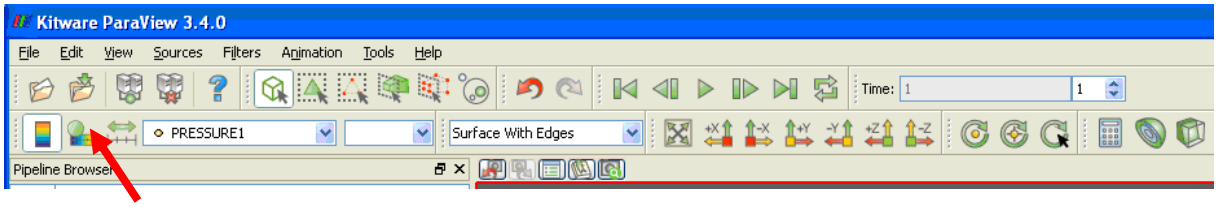


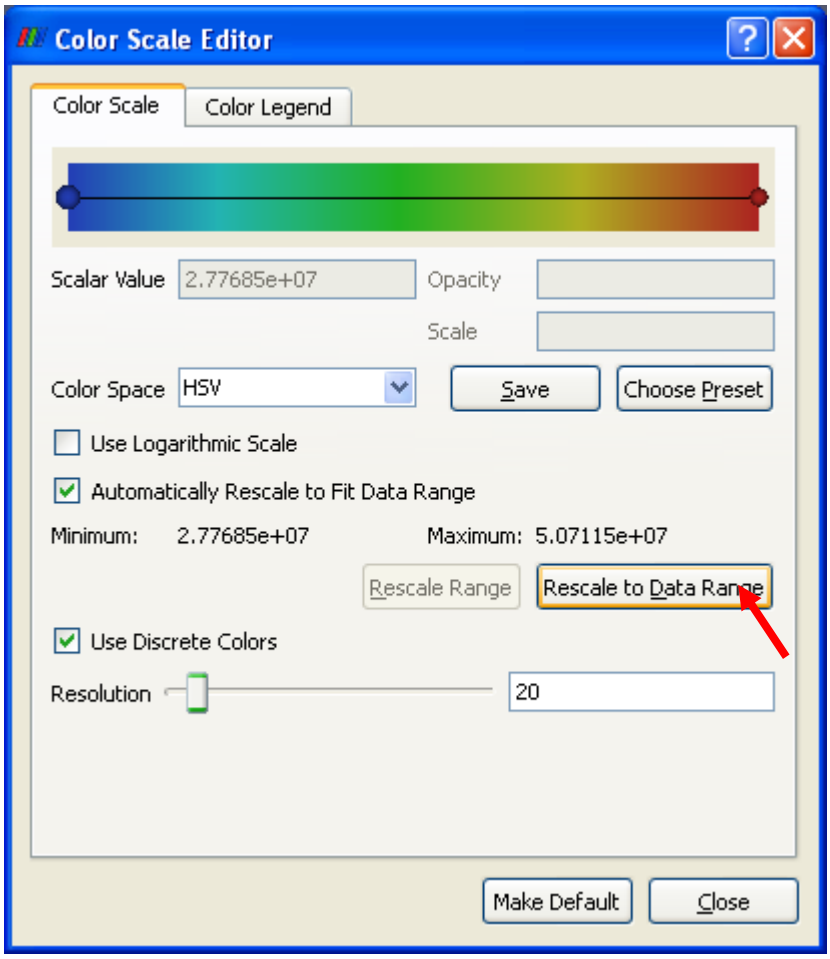


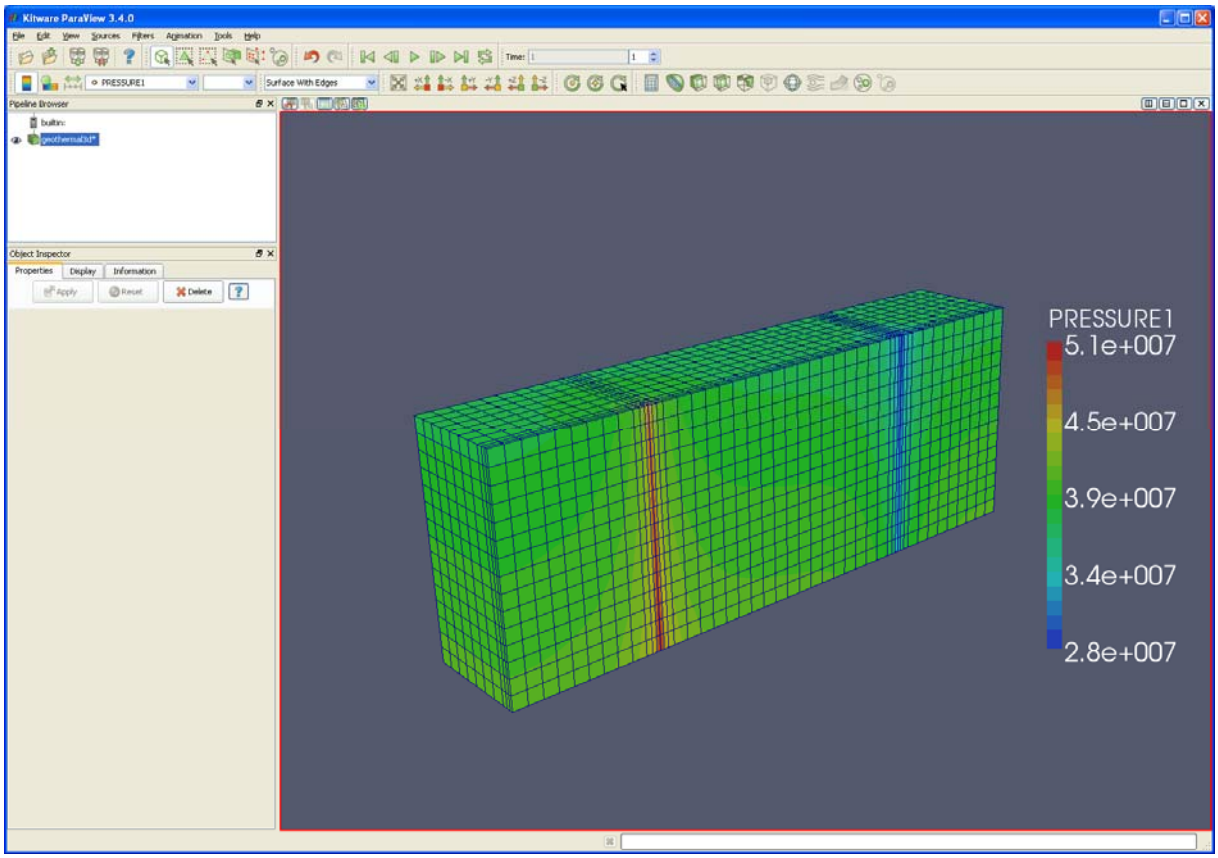
Show results of 1st time step.



Change color scale.




Color Scale Editor
Color Scale
Color Legend
Scalar Value
2.77685e+07
Opacity
Scale
Color Space
HSV
Save
Choose Preset
Use Logarithmic Scale
Automatically Rescale to Fit Data Range
Minimum:
2.77685e+07
Maximum:
5.07115e+07
Rescale Range
Rescale to Data Range
Use Discrete Colors
Resolution
20
Make Default
Close


PRESSURE1
5.1e+007
4.5e+007
3.9e+007
3.4e+007
2.8e+007

6 Simulation of heat transport (T-H coupled processes)

In this chapter, you simulate heat transport process with cold water injection for 10 years. A numerical model is built based on the previous model for hydraulic process. Initial conditions and boundary conditions are shown in Figure 5. Material properties have been already configures in the previous chapter.

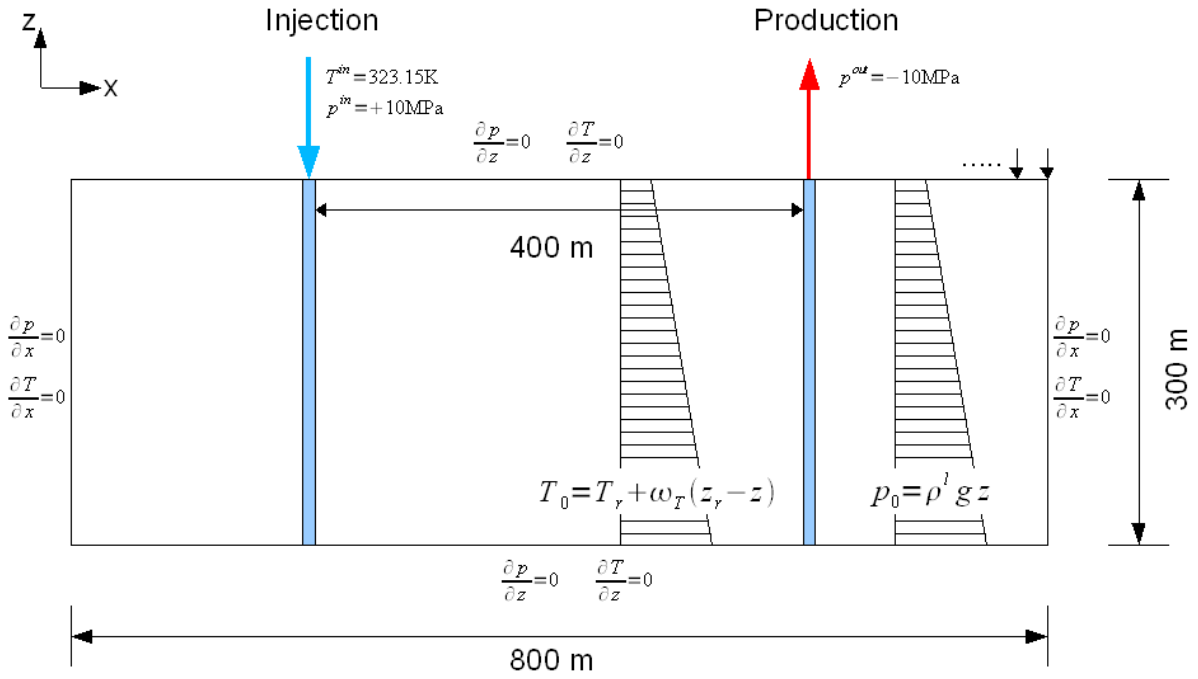
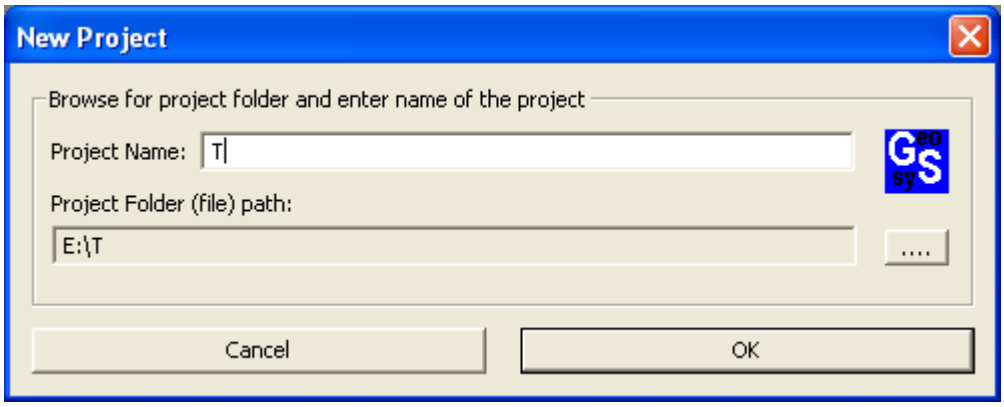


Figure 5. Numerical model for TH coupled processes

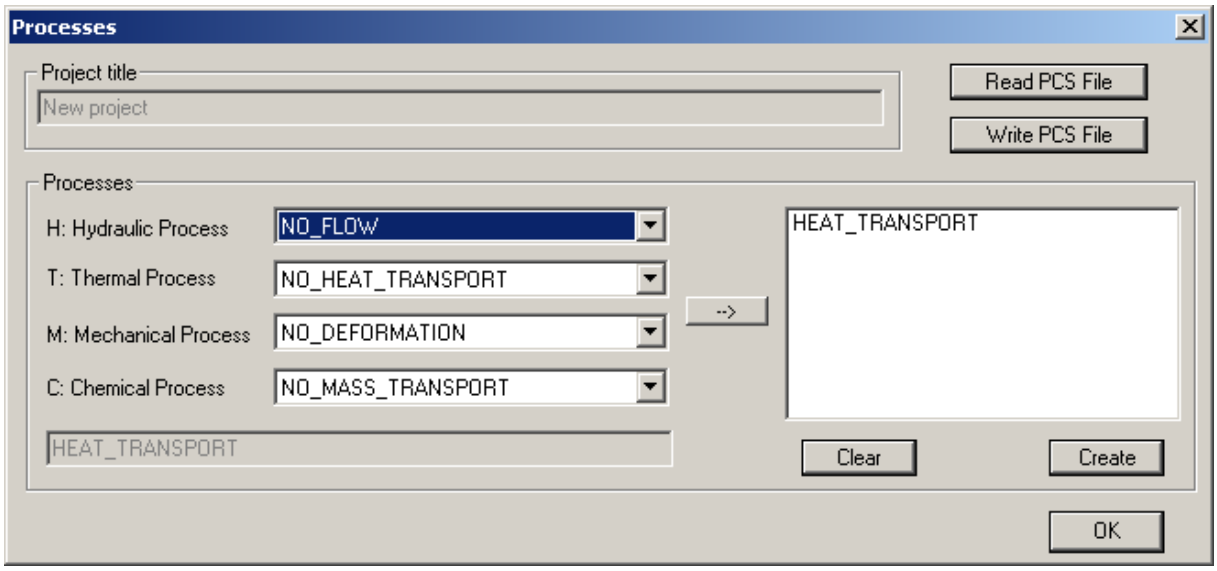
6.1 Creating HEAT_TRANSPORT process

Restart GeoSys. Create new GeoSys project as “T”.

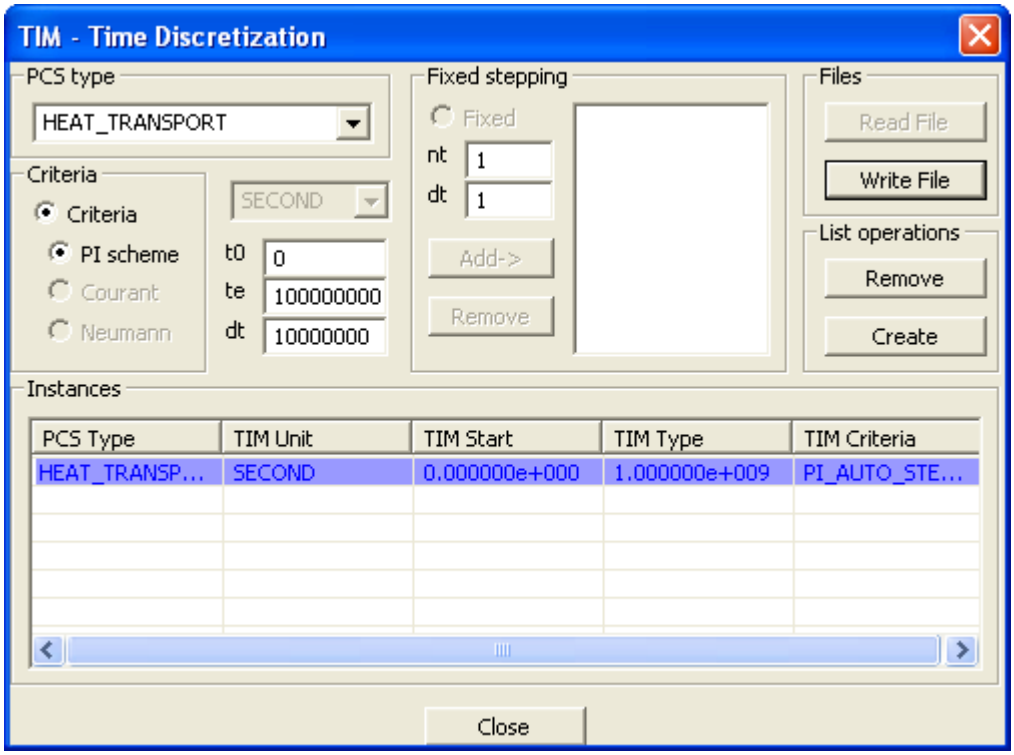


Import the GLI file and MSH file as you did for LIQUID_FLOW.

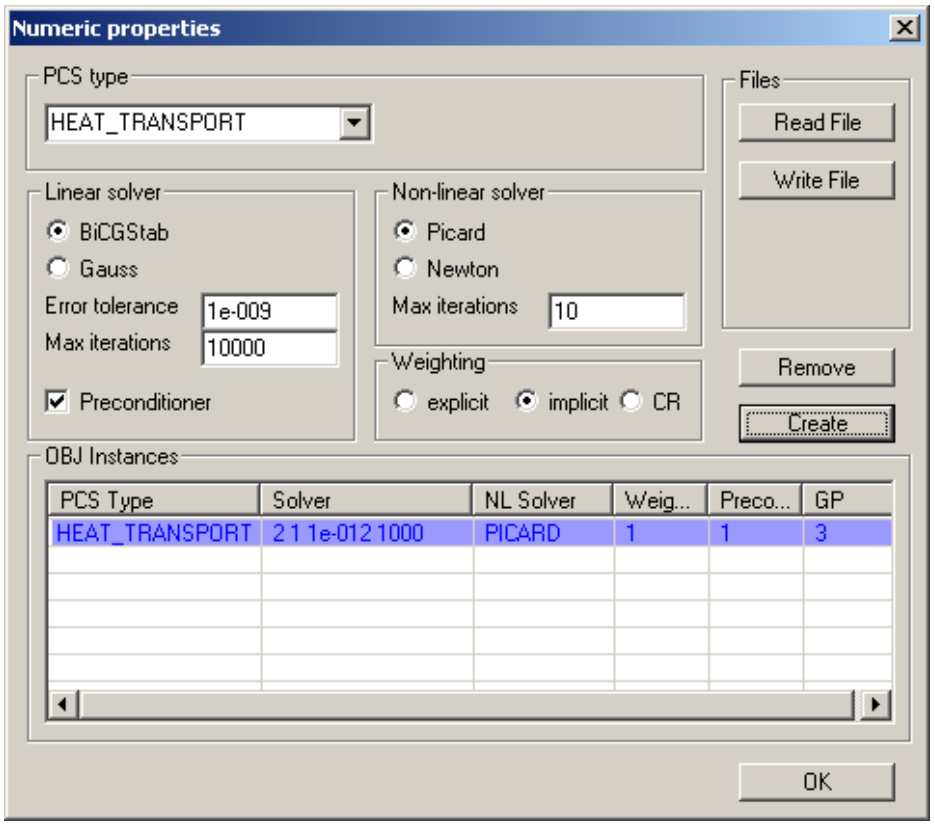
Click “PCS” on the PCS tab and add HEAT_TRANSPORT to processes.



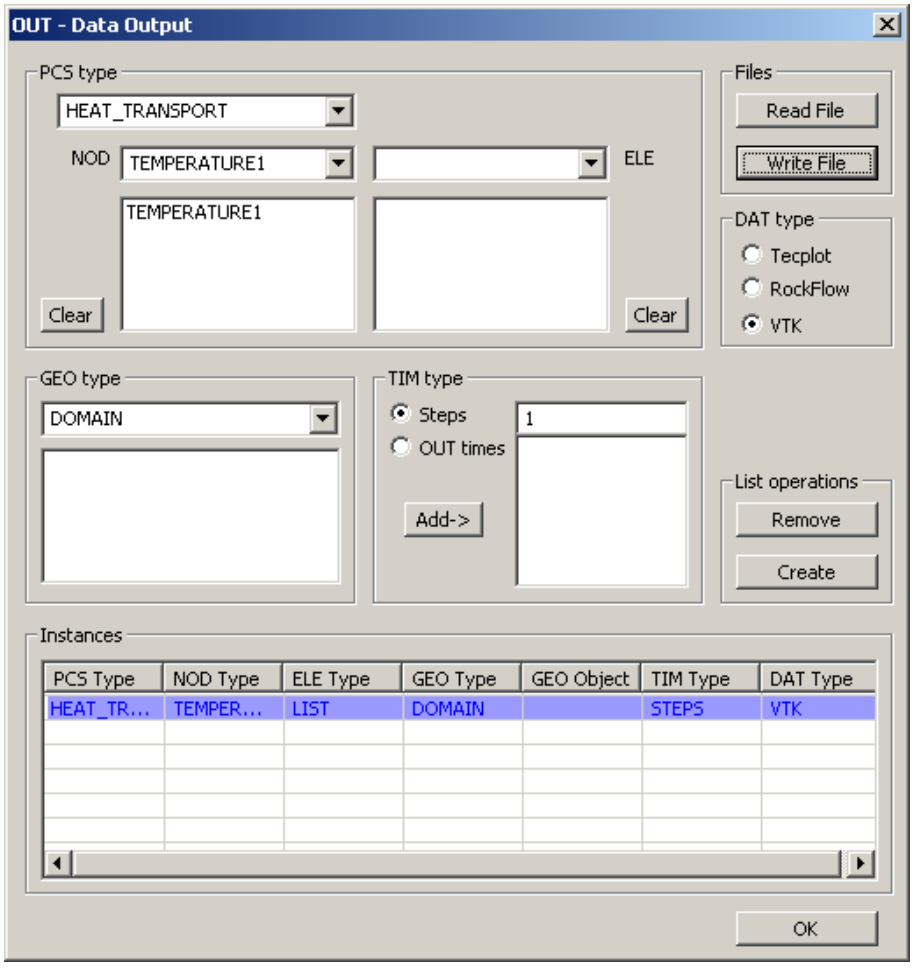
6.2 Time Discretization



6.3 Numerical properties

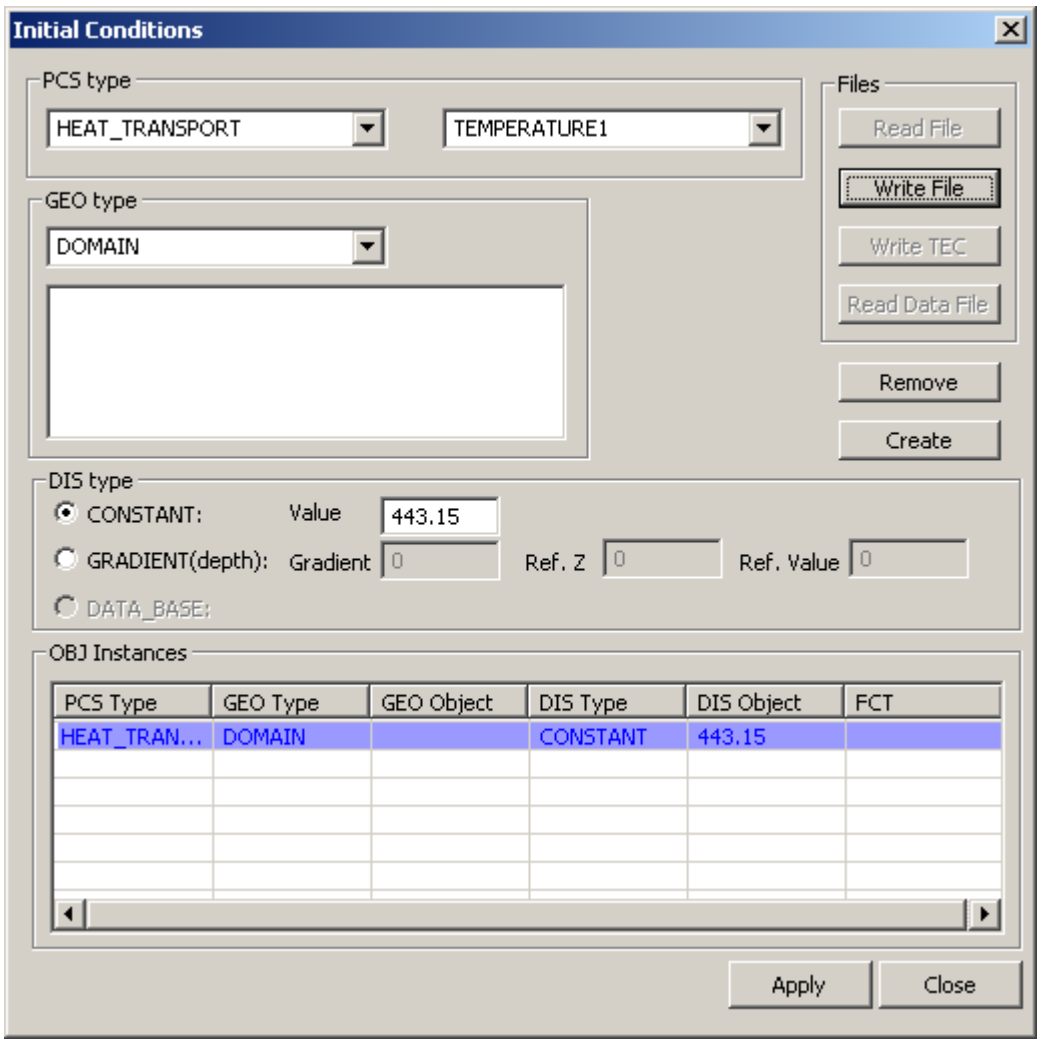


6.4 Data output



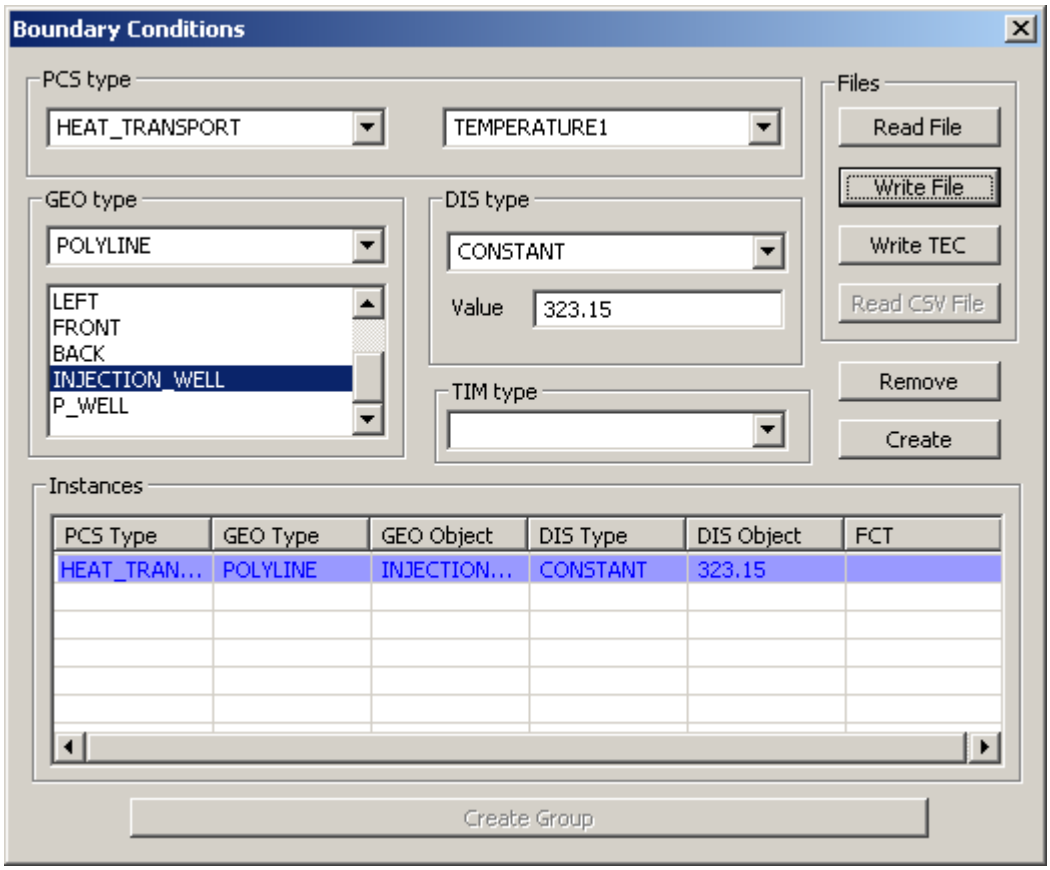
6.5 Initial Conditions

Click “IC” on the PCS tab. Enter initial conditions for heat transport as the below figure. Click “Create” button.
* In-situ temperature in the reservoir actually varies with the depth (Gradient is 0.029 K/m, reference temperature is 443.15K (170°C) at the depth of -4445.0m).



6.6 Boundary Conditions

Set temperature of 323.15K (50°C) at the injection well (POLYLINE: INJECTION_WELL).



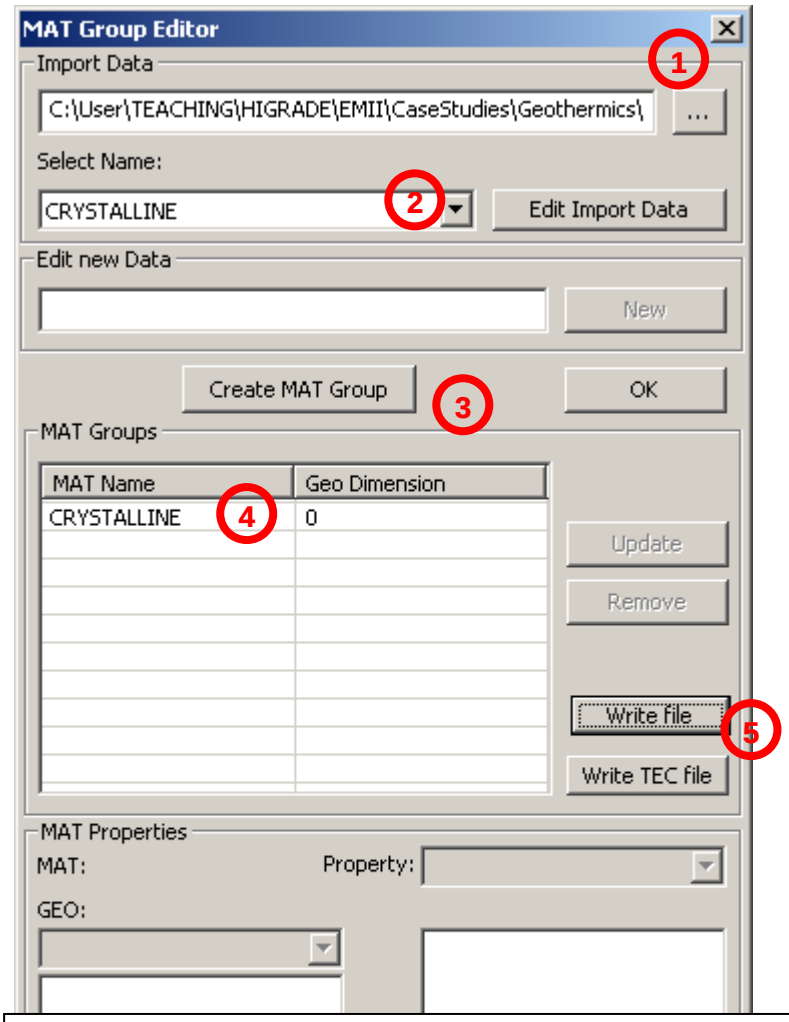
6.7 Fluid properties (MFP)

Creation of fluid material properties as in 5.7
or
Copy MFP file to the project.

6.8 Solid properties (MSP)

Click the MSP button on the PCS tab.

- Select the EXCEL file using the file browser (1).
- The available MSP groups are shown in the drop-down list (2)
- Select CRYSTALLINE and create MAT group (3)
- The selecetd group should appear in the MAT group list (4)
- Write the MMP file and save the OGS project (5)




```
GeoSys-MSP: Material Solid Properties -------------
#SOLID_PROPERTIES
 $NAME
  CRYSTALLINE
 $DENSITY
  1 2.850000000000e+003
 $ELASTICITY
  POISSION 2.500000000000e-002
  YOUNGS_MODULUS
  1 6.400000000000e+010
 $THERMAL
  EXPANSION
  1.000000000000e-004
  CAPACITY
  1 8.500000000000e+002
  CONDUCTIVITY
  1 2.790000000000e+000
#STOP
```

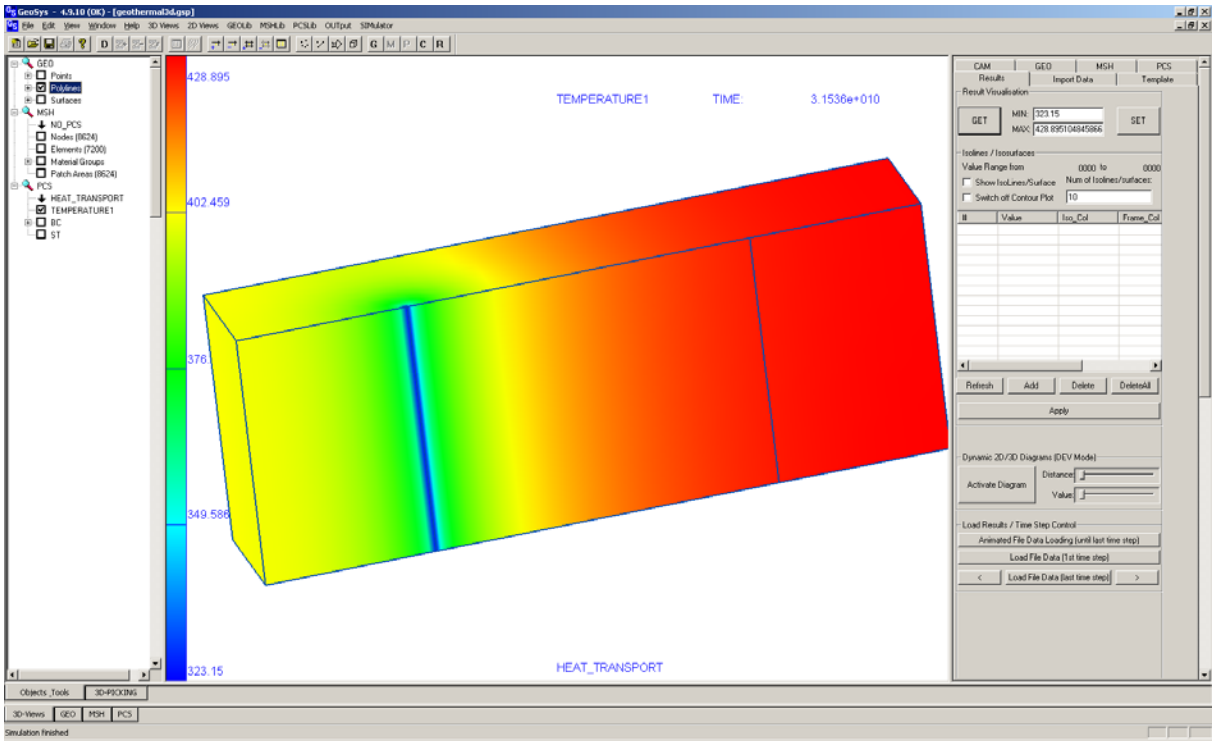
The MSP file should look like this. The specified solid properties are Young's modulus, Poisson ratio, thermal expansion coefficient and thermal properties.

6.9 Porous medium properties (MMP)

Import of porous medium properties as in 0
or
Copy MMP file to the project.

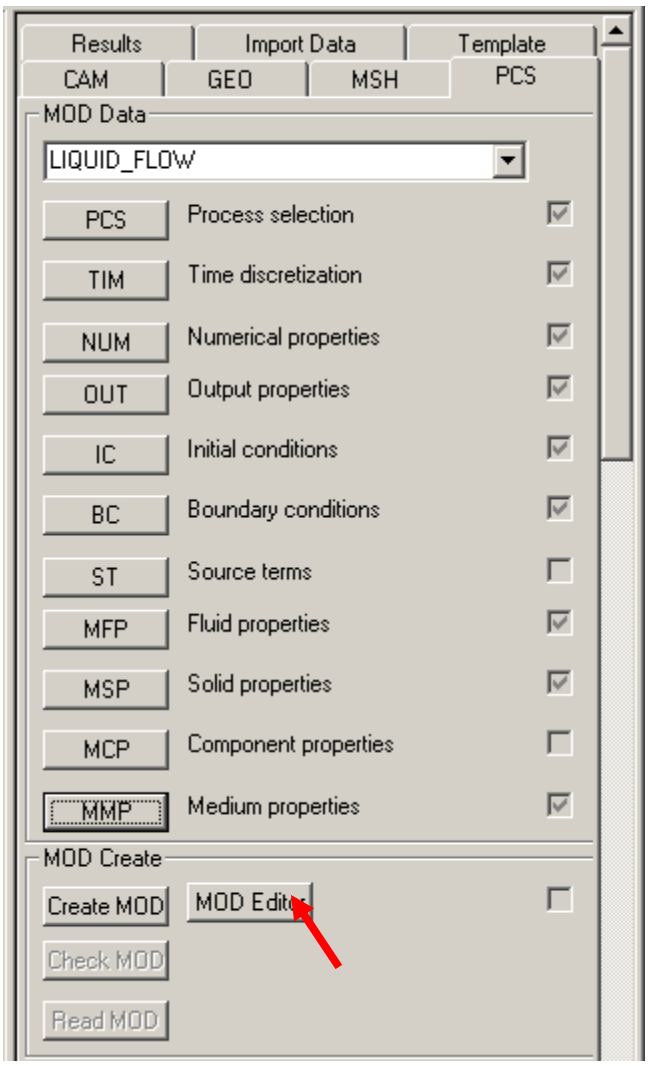
6.10 Run T simulation

Do not forget to create the MODel data before starting the simulation.



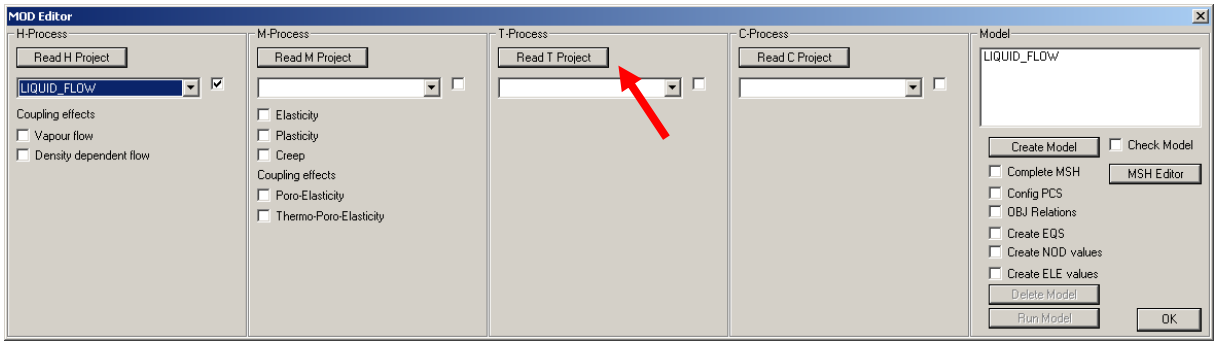
H and T have different time scales.

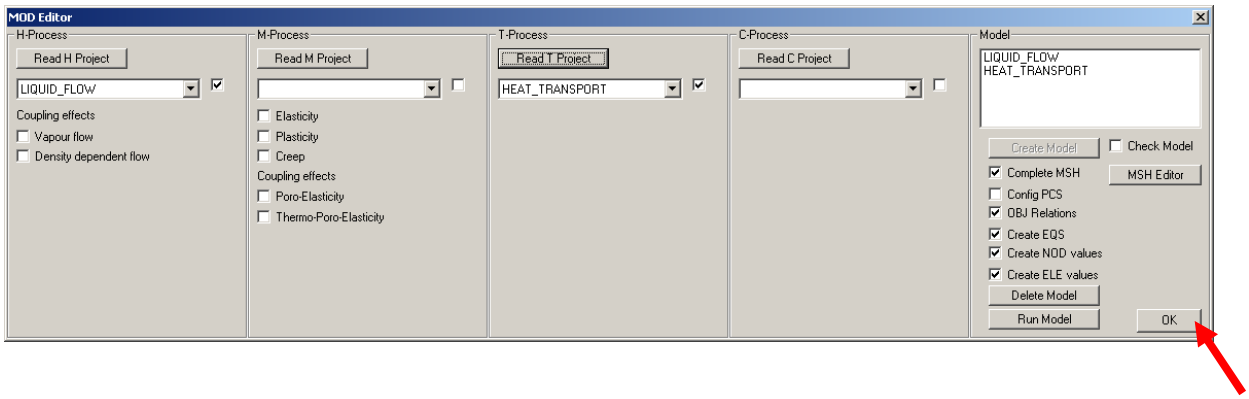
6.11 Coupling H and T Processes



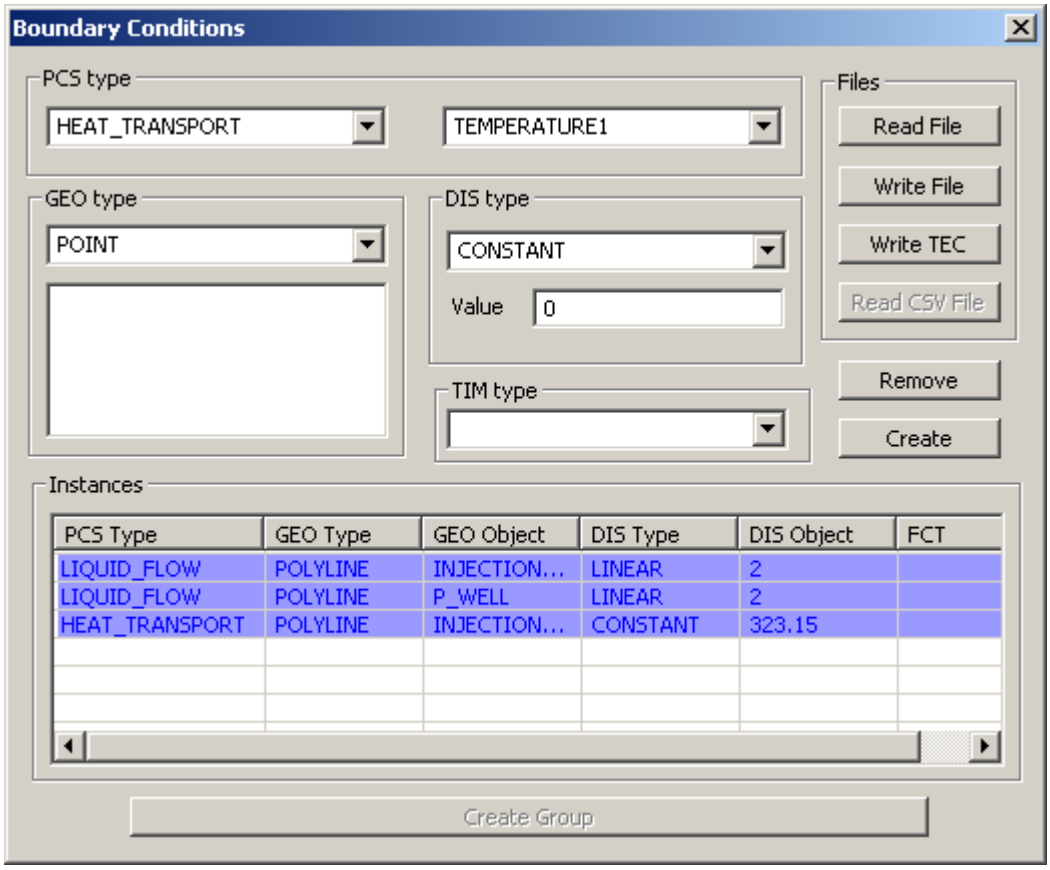
Now the H and T processes have been created and executed separately. Using the GeoSys Model Editor, processes can be combined. Before we do so please prepare a new folder for the coupled HT process and copy all input files for the H processes into this directory. Start GeoSys and load the H project from the HT folder.

After clicking the MOD Editor button the following dialog appears. As we started from the H model – a FLUID_FLOW process already exists. Now we can read existing T project and create the HT combined model and close the MOD editor.



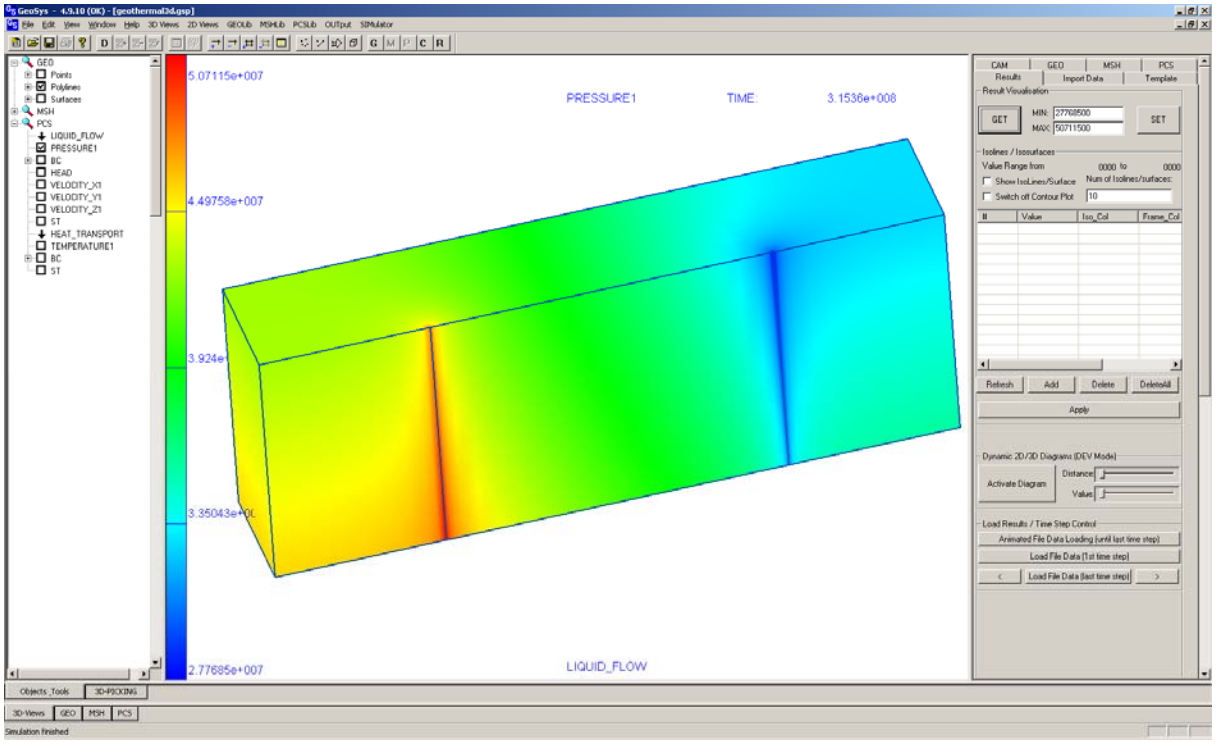


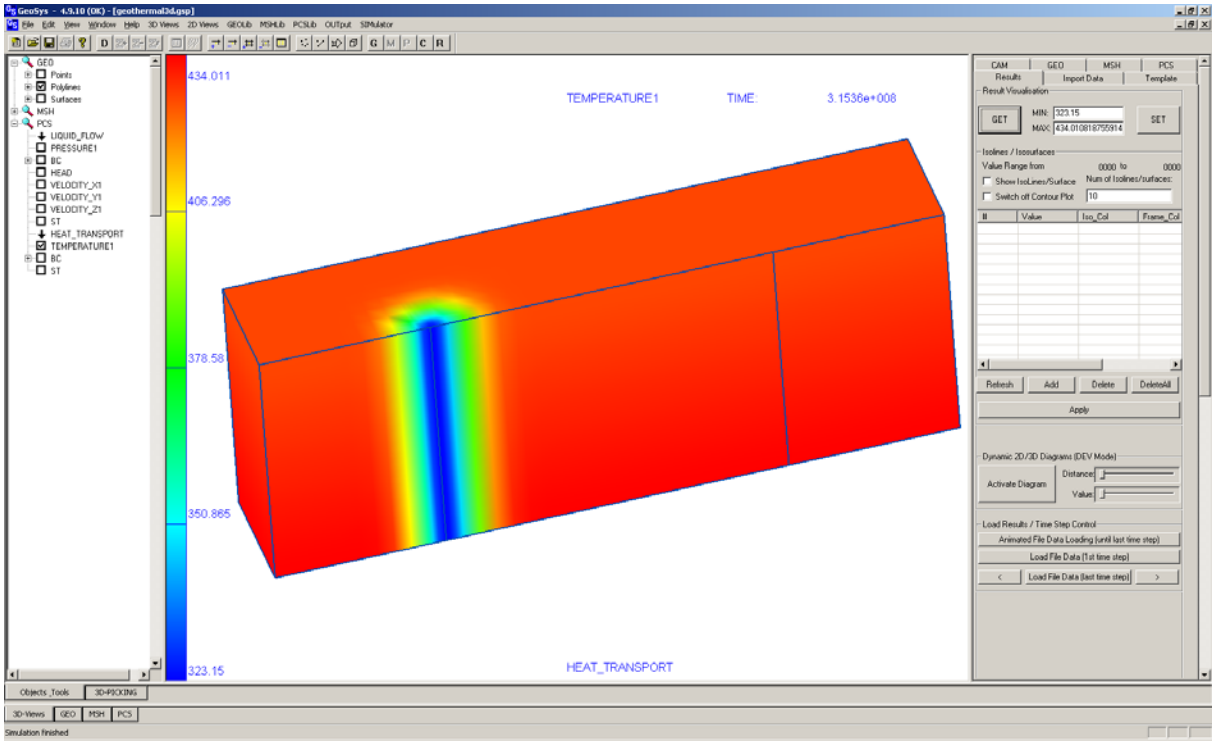
If you look now into the object data, you will the data from both the H and T processes, e.g. for boundary conditions



Save the combined project.

Now you can run the HT simulation.


5.07115e+007
4.49758e+007
3.924e+
3.35043e+0
2.77685e+007
PRESSURE1
TIME:
3.1536e+008
LIQUID_FLOW


434.011
406.296
378.58
350.865
323.15
TEMPERATURE1
TIME:
3.1536e+008
HEAT_TRANSPORT

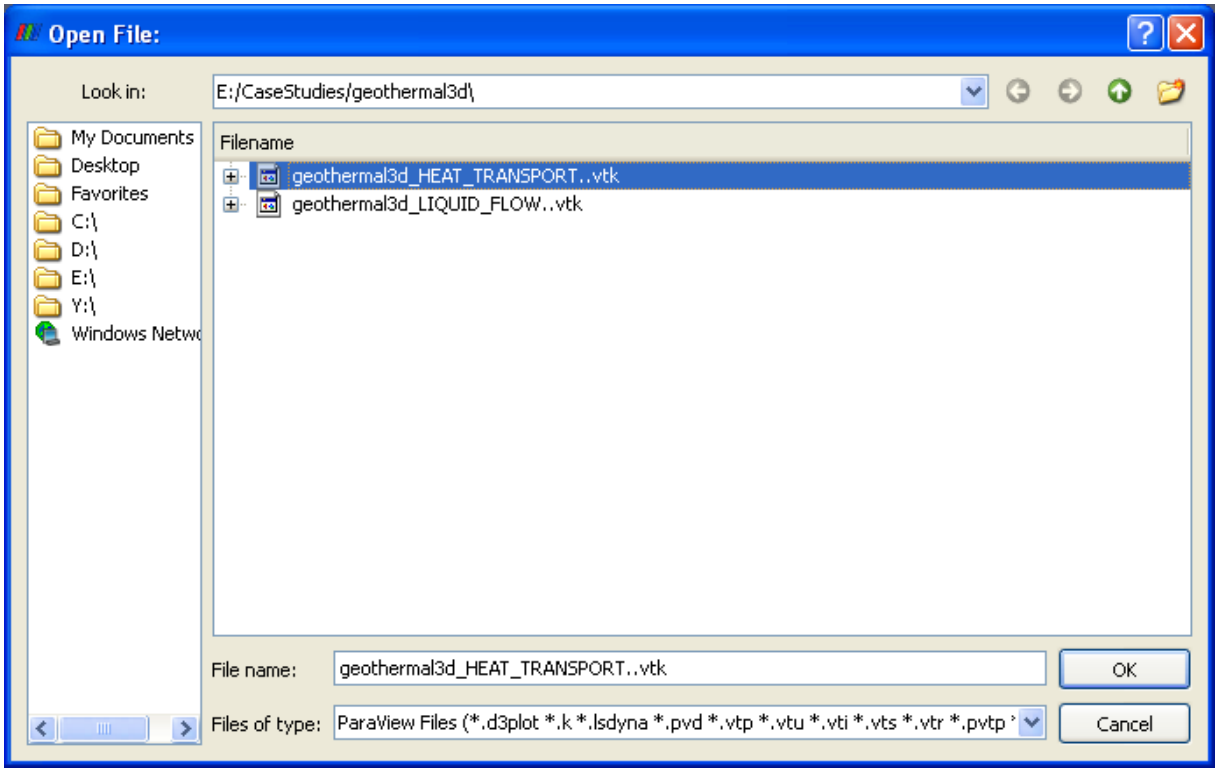
6.12 Visualize results

Check that two different series of VTK files have been created on the project folder after the HT simulation is finished,

- "geothermal3d_HEAT_TRANSPORT" + number + ".vtk"
- "geothermal3d_LIQUID_FLOW" + number + ".vtk".

Start ParaView and open VTK files.

Go to the file open dialog and select "geothermal3d_HEAT_TRANSPORT..vtk". Click OK.

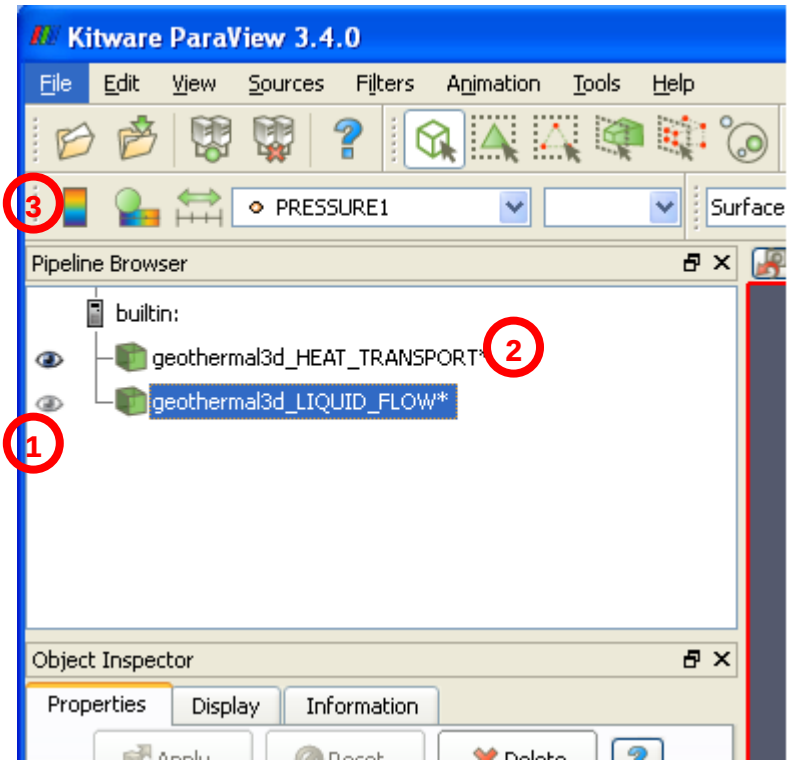


Click "Apply" button on the left "Object Inspector" view.

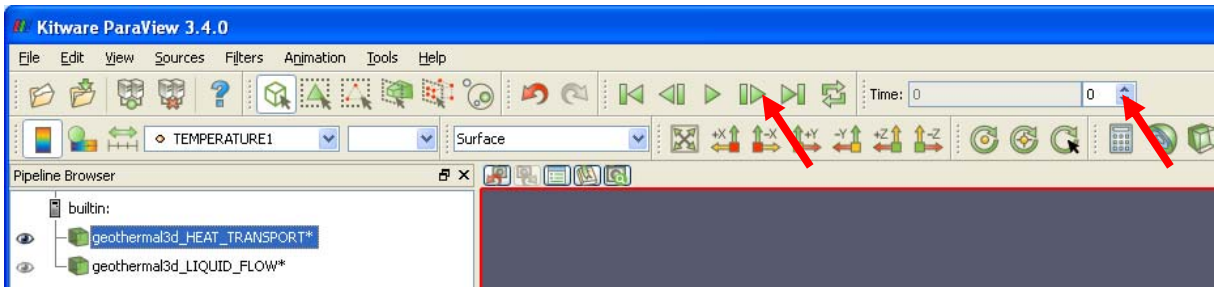
Go to the file open dialog again and select "geothermal3d_LIQUID_FLOW..vtk". Click OK. Click "Apply".

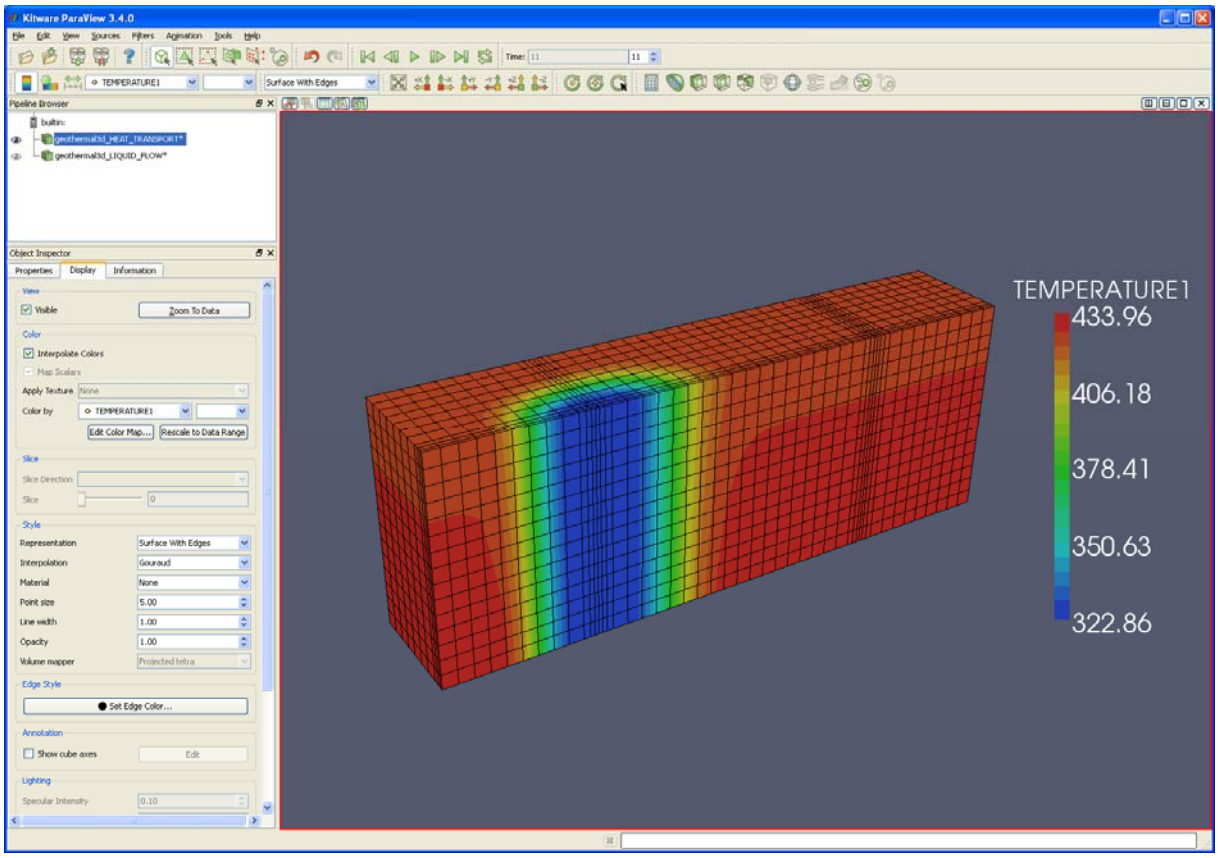
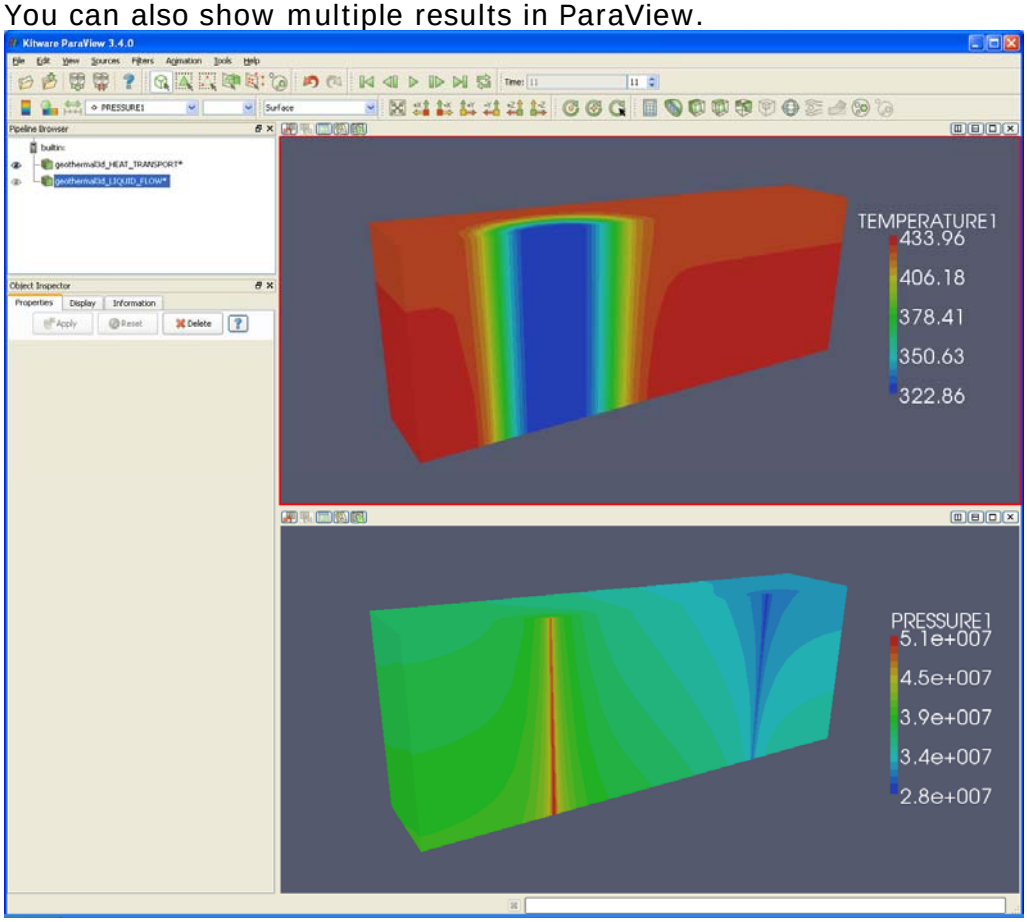
Show temperature field.

1. Click an eye mark of "geothermal3d_LIQUID_FLOW" in the Pipeline Browser.
2. Click "geothermal3d_HEAT_TRANSPORT" in the Pipeline Browser.
3. Show legend.



Change time step and adjust color scale if necessary.





You can also show multiple results in ParaView.