TOUGH2Viewer 2.0
Quick Tutorial
R1 (2020)

Stefano Bonduà, Villiam Bortolotti
DICAM

Università di Bologna

www.dicam.unibo.it

Summary

1 License agreement ... 3
2 Introduction ... 3
3 System Requirements ... 4
3.1 Windows Java3D installation troubleshot ... 4
4 Installation, execution and uninstallation instruction ... 6
4.1 TOUGH2Viewer installation ... 6
4.2 TOUGH2Viewer execution ... 8
4.2.1 Linux ... 8
4.2.2 Windows ... 8
4.3 TOUGH2Viewer uninstallation ... 8
5 TOUGH2Viewer overview ... 9
5.1 File menu ... 9
5.2 View menu ... 9
5.3 Analysis ... 10
5.4 Tools ... 11
6 Tutorial Examples ... 12
6.1 Structured grid model 2D ... 12
6.1.1 Load Files ... 12
6.1.2 3D Block Model view ... 13
6.1.3 2D Contour Plot visualization ... 14
6.1.4 2D Flow Vector visualization ... 15
6.1.5 Statistics ... 16
6.2 Structured 3D Grid model ... 18
6.2.1 Load Files ... 18
6.2.2 Export Structured Data to Paraview ... 18
6.2.3 3D block model view ... 19
6.2.4 3D Flow Vector visualization ... 25
6.2.5 3D Iso Surface ... 26
6.2.6 2D Contour Plot visualization ... 28
6.3 Unstructured 3D grid model ... 29
6.3.1 Load Files ... 30
6.3.2 3D Block model view ... 34
6.3.3 3D Flow Vector visualization ... 39
6.3.4 3D Iso Surface ... 40
6.3.5 2D Contour map ... 41
6.3.6 Statistics ... 42
6.4 Voronoi 3D grid model ... 43
6.4.1 Load Files ... 43
6.4.2 3D Block model view ... 44
6.4.3 Prepare GENER file ... 45
6.4.4 Export data to Paraview ... 49
6.5 The Amiata structured 3D grid case ... 50
6.6 The structured grid case ... 58
7 References ... 69
8 List of figures ... 70

1 License agreement

All computer programs (hereinafter collectively referred to as the 'Software') downloadable from the webpage https://github.com/stebond/TOUGH2ViewerX be used only under the following conditions:

2 Introduction

TOUGH2Viewer is a Java program capable of displaying unstructured (Voronoi complying) grids, locally refined and structured grids of TOUGH2 family of codes (Pruess et al. 1999). Structured grids are those complying with the MESHMAKER grid generator utility of TOUGH2.
In particular, TOUGH2Viewer allows to navigate through a 3D grid compatible with the TOUGH2 data file format and visualize:

- the thermodynamic variables and the material (namely, petrophysical properties) of each block;
- maps of isovalues (2D) of all thermodynamic variables;
- isosurfaces (3D) of all thermodynamic variables;
- flows of mass and heat between blocks;
- spatial profiles of thermodynamic variables along Cartesian direction;
- time plot of thermodynamic variables of a selected block.

All commands to manage a 3D visualization (zooming, pan, rotations) are CAD complying.
In the following chapters, four driven examples will be presented:

1. structured 2D grid generated with MESHMAKER.
2. structured 3D grid generated with MESHMAKER.
3. unstructured 3D (2.5D) grid generated with AMESH ().
4. Fully unstructured 3D grid generated using VORO2MESH (Bonduà et. Al, 2017)

All the example cases reported in this tutorial are simulation results obtained using the module EWASG (Battistelli et al., 1997) as implemented in iTOUGH2 (Pruess, 1999).
Related publications can be found in Bonduà et al. (2012, 2017), Berry et al. (2014).

3 System Requirements

The minimum system requirements to use TOUGH2Viewer are:
1GB of RAM;
100 MB of free disk space;
Graphic card equipped with 512 MB RAM;
Supported OS: Linux Ubuntu 8.04 or later, Windows XP, Vista, 7, 8 10.
TOUGH2Viewer requires **Java™ Runtime Environment (JRE)** version 1.6.14 (or later) and the Java extension to display three dimensional graphics (**Java3D**) version 1.5.1 (or later) to be already installed on the system. To download installers and instruction manuals, please visit the official Oracle Web Site.

3.1 Windows Java3D installation troubleshot

Some time the Java3D installation package cannot install properly the Java3D runtime libraries. Also, when the JRE is updated to new version, the Java3D package will stop to work. It is possible to fix this issue by the following procedure:

Open a File manager windows and locate the java folder. It will be in the "C:\Program Files\Java" folder as showed Fig. 1.

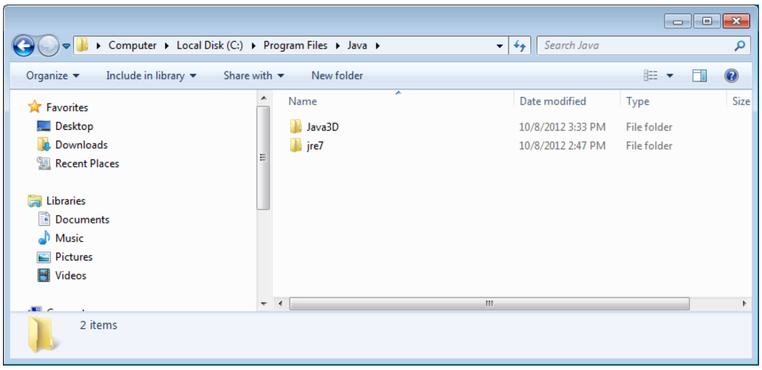


Fig. 1 – Java Folder

Note: if you install the JRE 32 bit version, you will find the Java folder located at "C:\Program Files (x86)\Java", and in the following instructions, you should use this path.

Copy the file "j3dcore-ogl.dll" from the folder "C:\Program Files\Java\Java3D\1.5.2\bin\" (Fig. 2) and paste it in the folder "C:\Program Files\Java\jre7\bin".

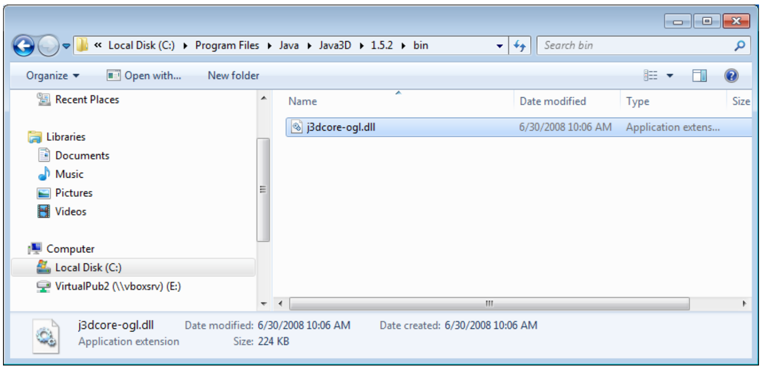


Fig. 2 – j3dcore-ogl.dll location

Copy the files j3dcore.jar, j3dutil.jar and vecmath.jar from folder

“C:\Program Files\Java\Java3D\1.5.2\lib\ext” (Fig. 3), and paste them in the folder

 “C:\Program Files\Java\jre7\lib\ext”

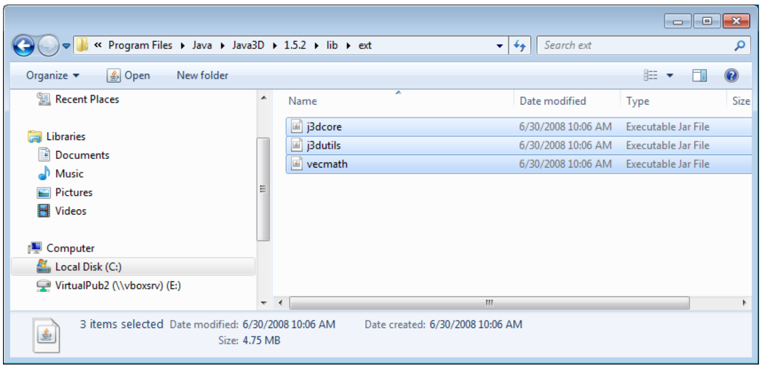


Fig. 3 – j3dcore.jar, j3dutil.jar and vecmath.jar folder

If you have more than one java installation folder, e.g. jdk1.6.x.y etc., repeat steps from “a” to “d” for each folder.

4 Installation, execution and uninstallation instruction

4.1 TOUGH2Viewer installation

TOUGH2Viewer bytecode can be downloaded from:
https://github.com/stebond/TOUGH2Viewer/blob/master/T2Viewer_v2.0.zip.
To install TOUGH2Viewer, user simply has to copy into a folder the compressed file (TOUGH2Viewer_v.X.zip where X indicates the version) containing the whole software programs and, once finished copying, unzip it. The unpacked package of files consists of the directory, **dist** (standing for distribution). The example data folder must be download separately from https://github.com/stebond/TOUGH2Viewer/blob/master/ExampleData.zip. In the examples described in this manual, the supposed folder will be "D:\T2Viewer\". So, at the end of the installation one should have a directory structure like the following:

D:\T2VIEWER

```
\---dist
    |  go.bat
    |  go1024.bat
    |  go1540.bat
    |  go2048.bat
    |  go3072.bat
    |  go4096.bat
    |  go6144.bat
    |  image.png
    |  iTough2Viewer.ini
    |  License.txt
    |  LogFile.txt
    |  path.ini
    |  README.TXT
    |  shape.ini
    |  splash.jpg
    |  tough2viewer.ini
    |  Tough2Viewer.jar
    |  voro_pp.ini
    |
    \---lib
          AbsoluteLayout.jar
          commons-math3-3.6.1.jar
          geoapi-2.3-M1.jar
          geoapi-pending-2.3-M1.jar
          gnujaxp.jar
          gt-api-2.7-M4.jar
          gt-coverage-2.7-M4.jar
          gt-data-2.7-M4.jar
          gt-geometry-2.7-M4.jar
          gt-jts-wrapper-2.7-M4.jar
          gt-main-2.7-M4.jar
          gt-metadata-2.7-M4.jar
          gt-referencing-2.7-M4.jar
          gt-referencing3D-2.7-M4.jar
          gt-render-2.7-M4.jar
          gt-shapefile-2.7-M4.jar
          gt-shapefile-renderer-2.7-M4.jar
          gt-swing-2.7-M4.jar
          iText-2.1.5.jar
          j3dcore.jar
          j3dutils.jar
          jcommon-1.0.16.jar
          jfreechart-1.0.13-experimental.jar
          jfreechart-1.0.13-swt.jar
          jfreechart-1.0.13.jar
          jsr-275-1.0-beta-2.jar
```

```
        jts-1.11.jar
        junit.jar
        miglayout-3.7-swing.jar
        netcdfAll.jar
        servlet.jar
        sgt-tutorial.jar
        swtgraphics2d.jar
        vecmath.jar
        visad.jar
        vtk.jar

ExampleData|
+---01_Structured_2D
|       2D.out
|       MESH
|
+---02_Structured_3D
|       3D_F-im-w.out
|       MESH.dat
|
+---03_Unstructured_3D
|   |   3d_unstruct_03.out
|   |   in.dat
|   |   segmt.dat
|   |
|   \---Maps
|       +---base_ss
|       |       base_ss.dbf
|       |       base_ss.prj
|       |       base_ss.qix
|       |       base_ss.shp
|       |       base_ss.shx
|       |       base_ss.xls
|       |
|       \---Raster
|               srtm_dem
|
+---04_Voronoi_3D
|       MESH
|       tough2viewer.dat
|       voronoi3D.out
|
+---05_Amiata3D
|       01dem.dat
|       01dem.dat.ply
|       02fractured.dat
|       02fractured.dat.ply
|       03caprock_shal.dat
|       03caprock_shal.dat.ply
|       04shallow_res.dat
|       04shallow_res.dat.ply
|       05caprock_deep.dat
|       05caprock_deep.dat.ply
|       06bottom-5000.dat
|       06bottom-5000.dat.ply
|       MESH
|       tough2viewer.dat
|
\---06_Structured2D
    +---model
    |       MESH
    |       tough2viewer.dat
    |       INCON.dat
    |
    \---Surfaces
            00top.dat
            00top.dat.ply
            01top.dat
            01top.dat.ply
```

02top.dat
02top.dat.ply
03top.dat
03top.dat.ply

4.2 TOUGH2Viewer execution

4.2.1 Linux

1. Navigate to the installation folder (/T2Viewer/dist/);
2. open a terminal window in “dist” folder
3. run the shell script, typing:

   ./**go.bat** [↵]

   or typing:

   java -Xmx512M -jar Tough2Viewer.jar [↵]

4.2.2 Windows

1. Navigate to the installation folder (/T2Viewer/dist/);
2. double click on file “**go.bat”** or open a DOS prompt and type:

   java -Xmx512M -jar Tough2Viewer.jar [↵]

4.3 TOUGH2Viewer uninstallation

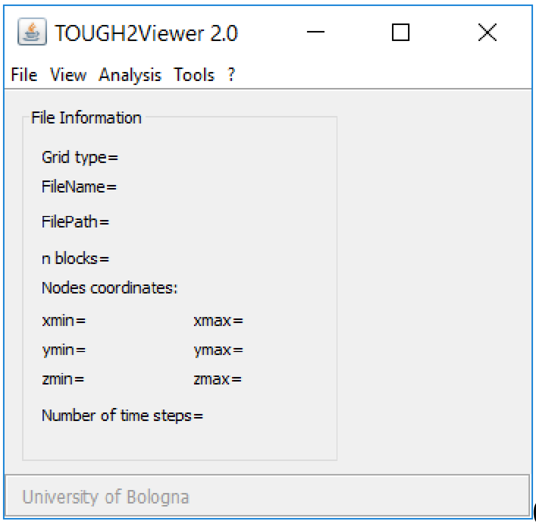
To remove TOUGH2Viewer, just delete the installation folder containing all the files. To obtain information about to remove JRE and Java3D, consult the JRE and Java3D manuals or visit the Oracle official web site.

5 TOUGH2Viewer overview

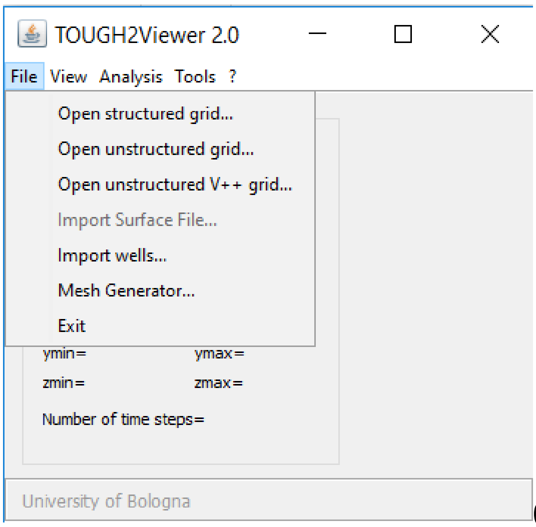
TOUGH2viewer is a dialog based GUI for visualization and editing of TOUGH grids and simulation results.

The main menu of TOUGH2Viewer contains the following items (Fig. 4 (a)):

- File: read grid and simulation results files;
- View: grid visualization;
- Analysis: results visualizations;
- Tools: visualization options and editing tools;
- ?: information about TOUGH2Viewer version.



(a)



(b)

Fig. 4 – (a) TOUGH2Viewer main dialog (b) File menu items.

5.1 File menu

The File menu Fig. 4 (b) allow the reading of the several file format of TOUGH grids.
File menu items are:

- Open structured grid: imports grid generated by MESHMAKER;
- Open unstructured grid: imports grid generated by AMESH;
- Open unstructured V++ grid: imports grid generated by VORO2MESH;
- Import Surface files: Import surface files (grid type or PLY file format);
- Import wells and Mesh Generator are under development.

5.2 View menu

The **View** menu Fig. 5 allow grid and simulation results visualizations.
View menu items are:

- **3D Block model**: 3D grid visualization;
- **3D Flow vector**: 3d flow visualization;
- **3D Iso Surface**: Iso Surface generation;
- **2D Contour plots**: contour plot of cross section of the model
- **2D Flow vector**: flow visualization for 2D grids.

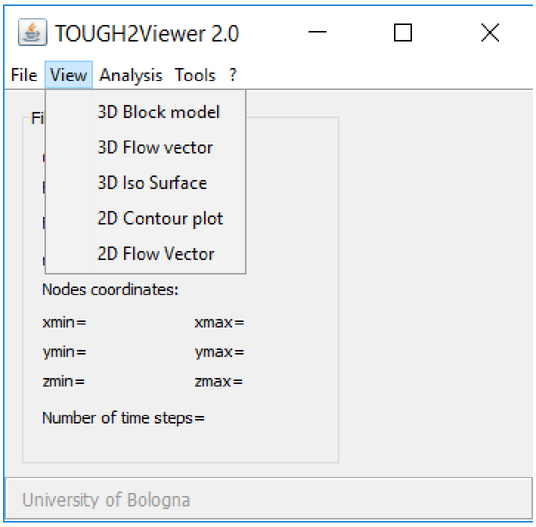


Fig. 5 – TOUGH2Viewer View menu

5.3 Analysis

The Analysis meu allow to obtain statistic and variables plots of the simulation results and contains the following items:

- **Statistics**: global statistics of simulation results;
- **Heat and Mass Balance**: global behaviour of the variable plotted by the BALLA function of TOUGH as read from the simulation results;
- **Plot variables**: allow plotting of variables with one or two range axis.

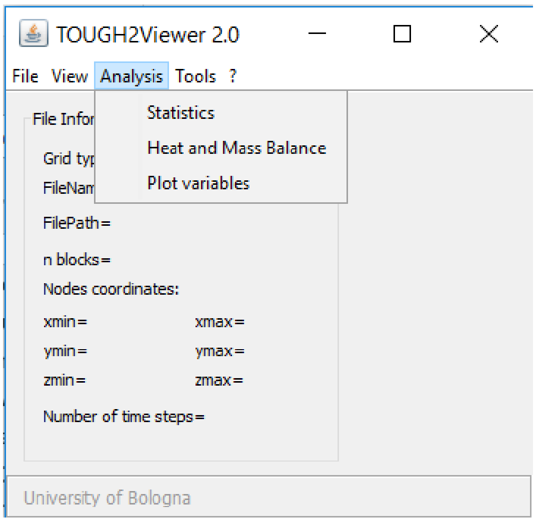


Fig. 6 – TOUGH2Viewer Analysis menu

5.4 Tools

The Tool menu contains the following items (Fig. 7):

- **Options**: visualizations settings and visualization options;
- **Color scale**: allow the preview and file export of the color scale;
- **Test Java3D**: test the correct Java3D installation;
- **Create INCON**: allow to generate the INCON file as specified by the user.

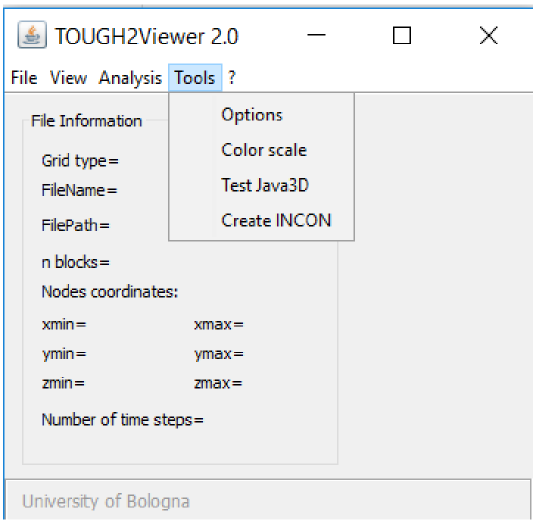


Fig. 7 – TOUGH2Viewer Tools menu

6 Tutorial Examples

These tutorials are designed to introduce the TOUGH2Viewer basic features, and should take less than an hour to be completed at all. Once finished, the user will have the needed skills to explore a 3D simulated model, plot variables and add to the 3D model Frame own maps.
The following examples refer to using TOUGH2Viewer under Windows operating system.

6.1 Structured grid model 2D

This example is related to the visualization of a 2D grid (50×1×36 number of blocks). The files of this tutorial are in the path: “D:\T2Viewer\ExampleData\01_Structured_2D\”.
The files are:

- MESH ->the classical MESH file of TOUGH
- 2D.out->simulation results of a numerical model computed using EWASG.

Start TOUGH2Viewer, the main window of TOUGH2Viewer will be displayed, as shown in Fig. 8.

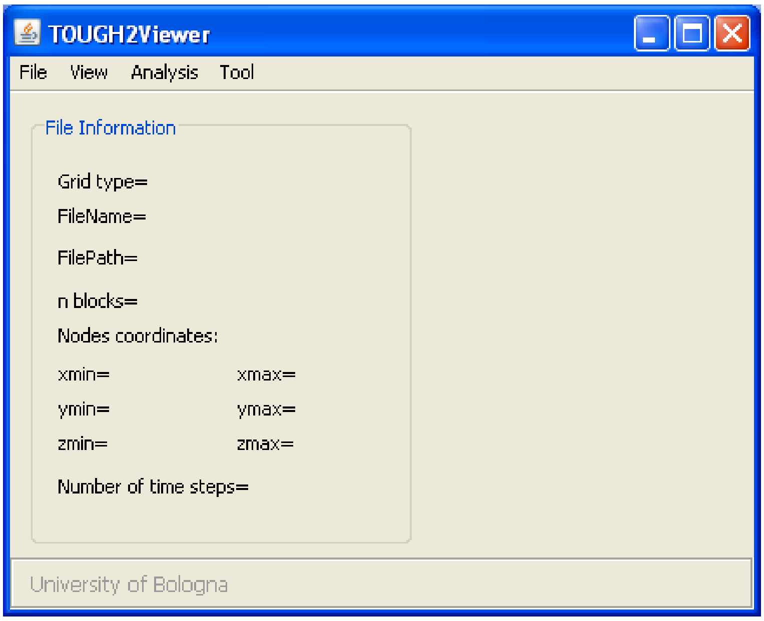


Fig. 8 - The TOUGH2Viewer empty main window

6.1.1 Load Files

From the main menu, select the **File->Open structured grid** menu item to open the "Open structured grid" dialog window (Fig. 9).

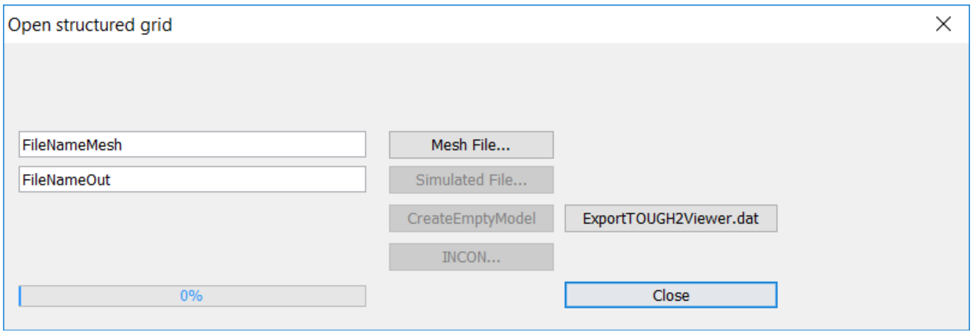


Fig. 9 - The “Open structured grid” dialog window.

Click the **Mesh File** button to display the "Open mesh" dialog window. Browse and select the “MESH.dat” file (\T2Viewer\ExampleData\01_Structured_2D\MESH.dat).
Click the Open button to load this file. This operation takes few seconds .When loading is finished, the text box on the left side of the **Mesh File** button will contain the full path of the mesh file.
Click the **Simulated File** button to display the "Open Simulation" dialog window. Locate the TOUGH2 simulation output file “2d.out” and select it by clicking on it.
Then, click the Open button to start loading the file. The dialog progress bar will show the loading process. To close the dialog and return to the main window, click **Close** button.
Now in the main window will be shown some geometric information about the model loaded (see Fig. 10).

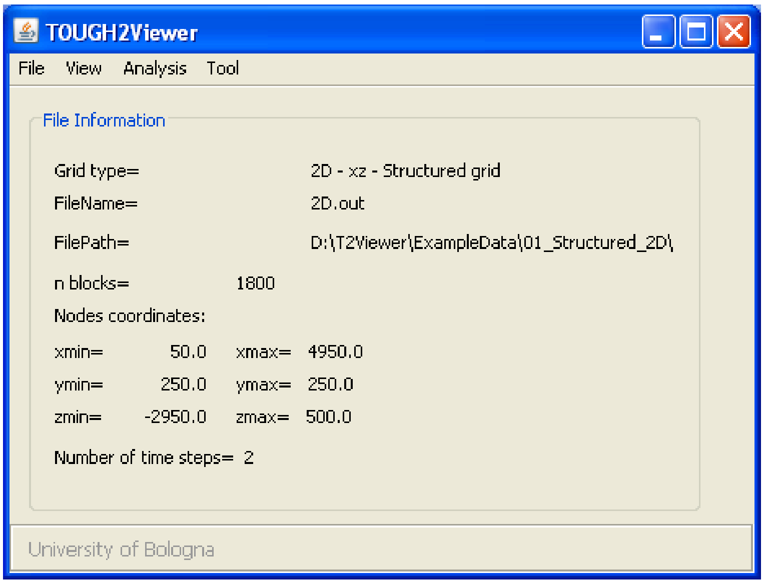


Fig. 10 - Main window with geometric information of the loaded grid.

6.1.2 3D Block Model view

In main menu, click the **View->3D Block Model** menu item, to display the "3D Block Model" dialog window and to have the default 3D Model Frame view (in this specific case only with one block in the y coordinate) of the grid, as shown in Fig. 11. The default variable showed in the dialog window is the first variable reported in the simulation output file (P for pressure in Fig. 4). Default action is **ShiftXY**.

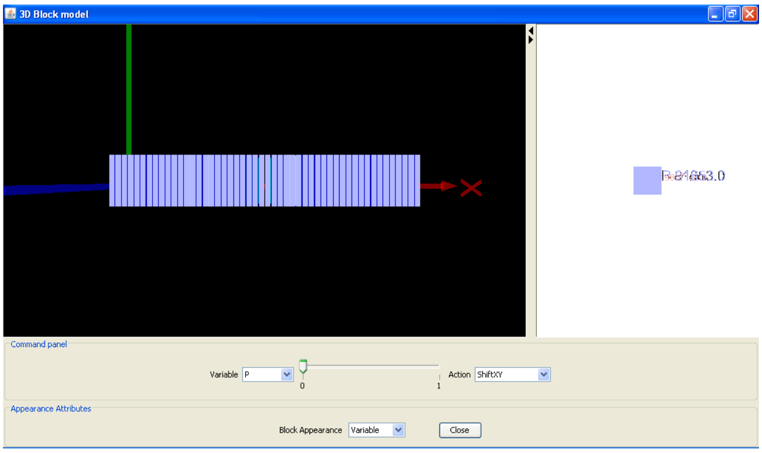


Fig. 11 - 3D Default representation of the Model, Top View.

From **Action** combo box, select “FrontView” to change view and then the plot frame is updated as shown in Fig. 5.

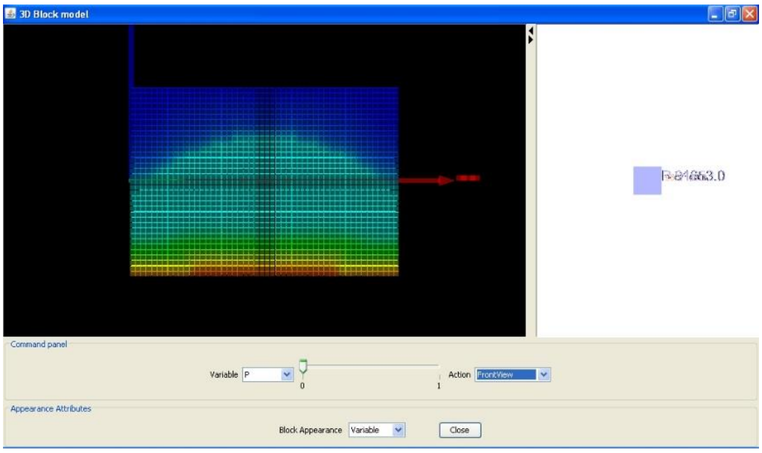


Fig. 12 - 3D representation of the model, FrontView.

Close the 3D Block model dialog by clicking the window icon or the **Close** button.
For more detail on commands available for the 3D visualization management, see the example in chapter 6.2 and 6.3.

6.1.3 2D Contour Plot visualization

This functionality enables to plot the primary variables by means of contour lines.
In the main menu, click the **View->2D Contour plot** menu item and the default "2D Contour plot" dialog window displays. From the Variable combo box, select **P** (Pressure), then click the **Plot** button (Fig. 13).

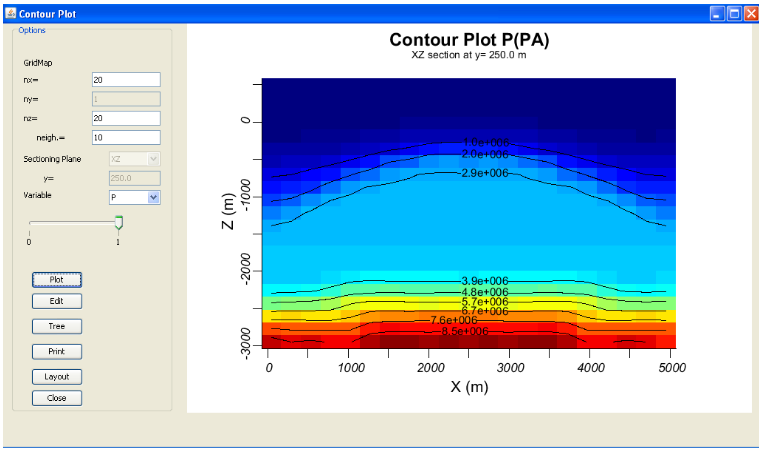


Fig. 13 - Pressure contour map

If one modifies *plot's* parameter, click the **Plot** button to update the plot. In the right part of the window (Fig. 14) is shown the contour plot of the temperature variable.

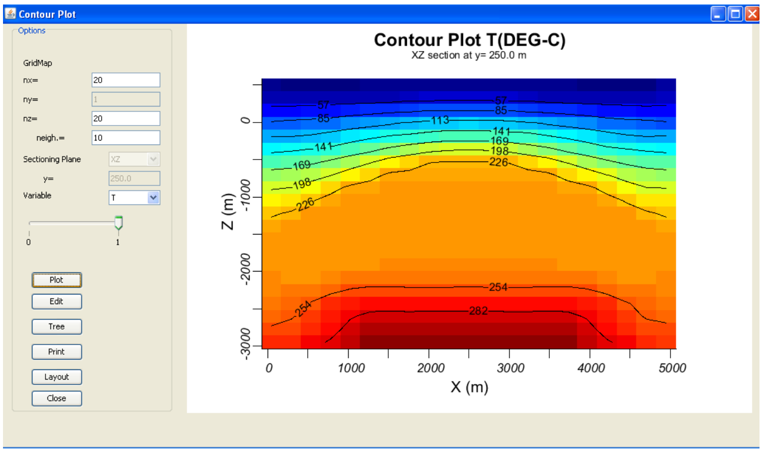


Fig. 14 - Temperature contour map

Close the 2D Contour plot dialog by clicking the window icon  or the **Close** button.

6.1.4 2D Flow Vector visualization

This functionality gives a 2D vector representation of the flow heat or mass exchange among blocks. In the main menu of TOUGH2Viewer, click the **View->2D Flow Vector** menu item to display the 2D Flow plot dialog window (see Fig. 8). The default plot uses the first variable present in the simulation output file, and the first time step (FHEAT in the figure).

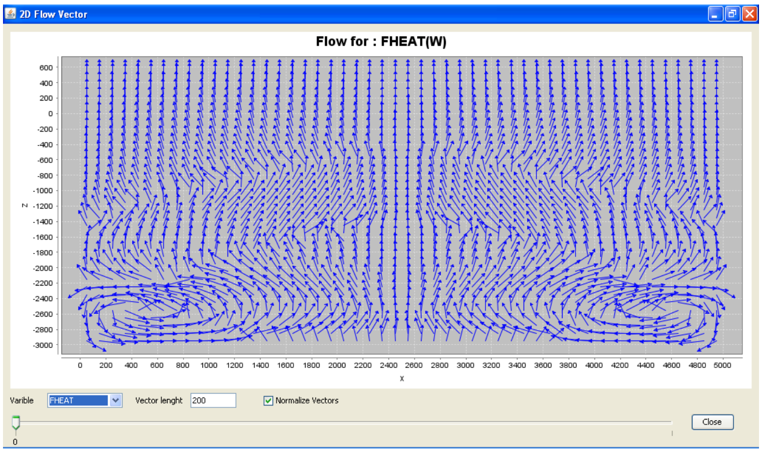


Fig. 15 - Vector map of the heat flow.

From the Variable combo box, select the VEL(LIQ.) item. Move the time slider to the last time step. Check the Normalize vectors check box and type 100 in to Vector length text box. The 2D Flow vector dialog window now is as shown in Fig. 16.

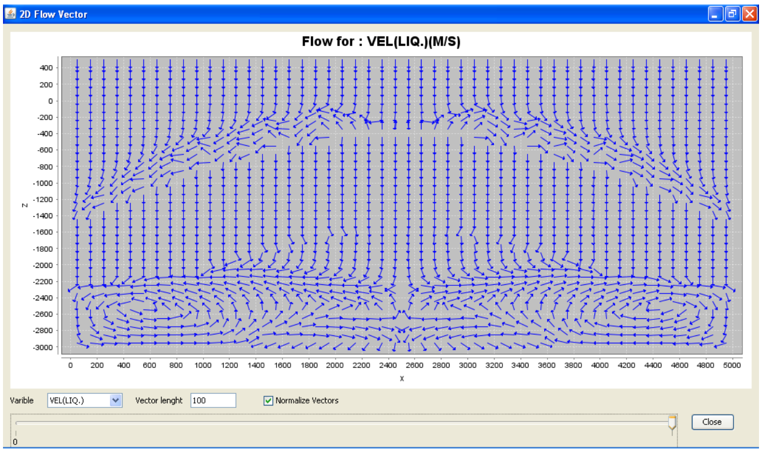


Fig. 16 - Vector map of the variable VEL(LIQ.).

Close the 2D Flow vector dialog by clicking the window icon  or the Close button.

6.1.5 Statistics

In the “Statistics” dialog window is shown a table with the maximum and the minimum value for all the thermodynamic variables, and it is possible to plot a variable vs simulated time steps.
From the menu bar, click the **Analysis->Statistics** menu item. The Statistics dialog window is displayed (Fig. 17).

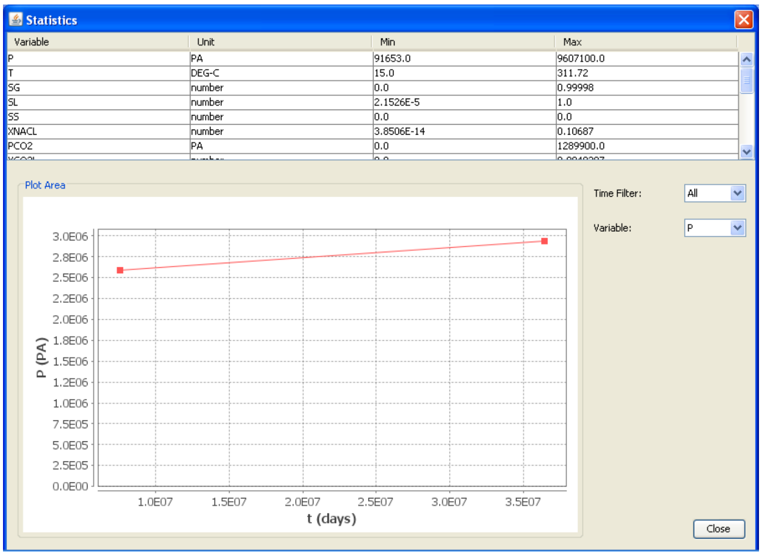


Fig. 17 - Min and Max value for simulation data. The default plot is first variable vs time.

To change the time steps, use the "Time" combo box. To change the plotted variable, use the “Variable” combo box.

Close the Statistic dialog by clicking the window icon  or the **Close** button.

Exit from TOUGH2Viewer by clicking the window icon  or by the main menu item **File->Exit**.

6.2 Structured 3D Grid model

This example manages a 3D model representing a spatial domain of 5000x5000x3600 m^3 discretized in 10×10×36 blocks.

- MESH ->the classical MESH file of TOUGH
- 3D_F-im-w.out.out->simulation results of a numerical model computed using EWASG.

The simulation results contain two time steps.
Start TOUGH2Viewer (under Windows OS) by double clicking the **go.bat** batch file.

6.2.1 Load Files

From the TOUGH2Viewer main menu, click the **File->Open structured grid** menu item, and then the "Open structured grid" is displayed (see Fig. 18).

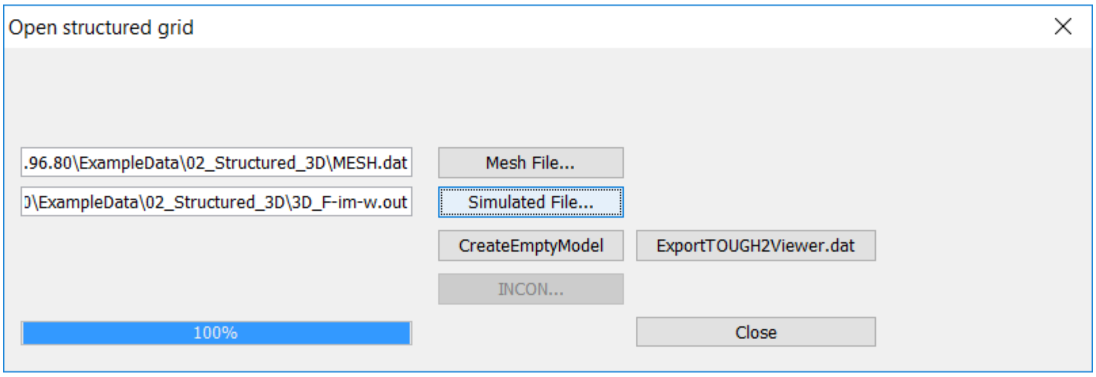


Fig. 18 - Open Structured grid dialog.

Click the **Mesh File** button to display the Open mesh dialog window and load the “MESH.dat” from the “\T2Viewer\ExampleData\02_Structured_3D\” folder.
Click the Open button to start loading the file. When loading is finished, the text box on the left side of the **Mesh File** button will contain the full path of the mesh file.
Click the **Simulated File** button, and then the **Open Simulated** dialog window is displayed. Select the “3D_F-im-w.out” file and click the **Open** button to load it. The progress bar of the dialog will show the loading advancement.
When the reading is complete, close the dialog by clicking **Close** button. Then, in the main window are shown some geometric information about the model.

6.2.2 Export Structured Data to Paraview[1]

Exporting the MESH and the simulation results to Paraview is quite simple using TOUGH2Viewer. After loading the MESH file and the simulation results of the 2D example, click the button **ExportTOUGH2Viewer.dat**. A file named *tough2viewer.dat* is created in same working folder. Close TOUGH2Viewer and re-start a new session of TOUGH2Viewer. Now choose from the menu the command **File->Open unstructured V++ grid …** . Using buttons, load the tough2viewer.dat file, the MESH and the simulation results. Click the **Export2VTU** button. A set of out_[i].vtu files is generated in the working folder. You can now open these files with Paraview. See also the chapter 6.4.4 for details.

[1] http://www.paraview.org.

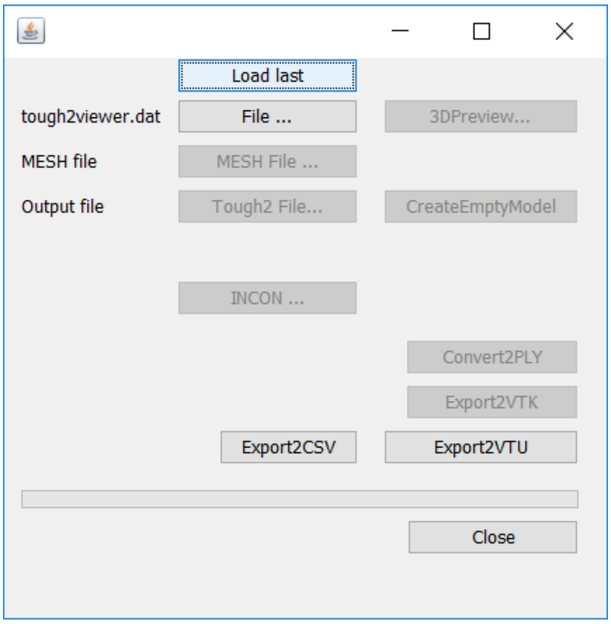


Fig. 19 - Open unstructured V++ grid dialog window.

6.2.3 3D block model view

From the main menu, click **View -> 3D block model** menu item. The "3D Block model" dialog window is displayed with the default setting, as shown in Fig. 20.

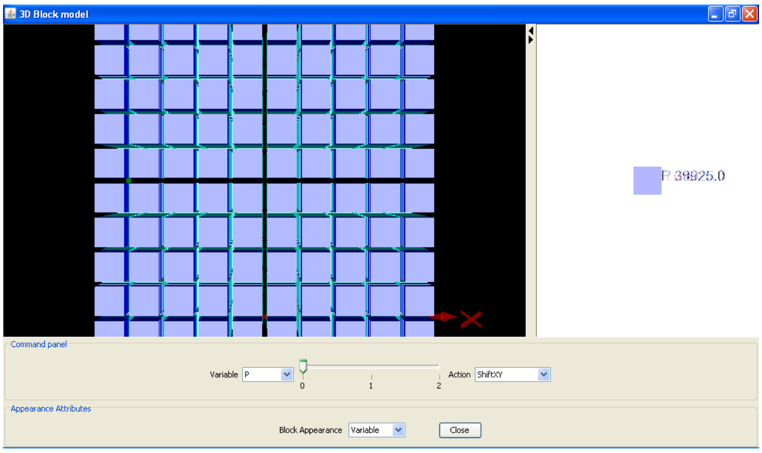


Fig. 20 - 3D Block model window, top view.

From the “Variable” combo box, select the **T** variable (Temperature). Move the time step slider in position 2. Rotate both the model and the color scale.
The plot of “3D Block model” dialog window, will be similar to that show in Fig. 21.

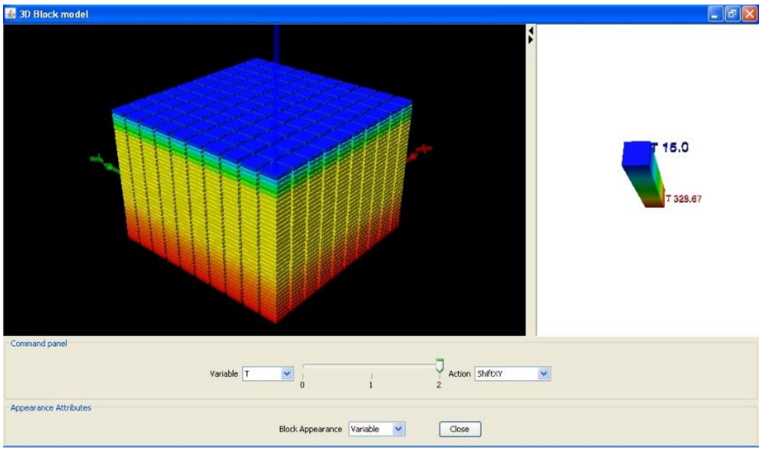


Fig. 21 - 3D Block model box showing the Temperature variable at the time step 2.

To split the 3D model Frame, from the “Action” combo box select the **ShiftXY** command, move the mouse on the 3D model frame and left click over a block. The upper part of the model above the clicked block will be shifted as shown in Fig. 22.

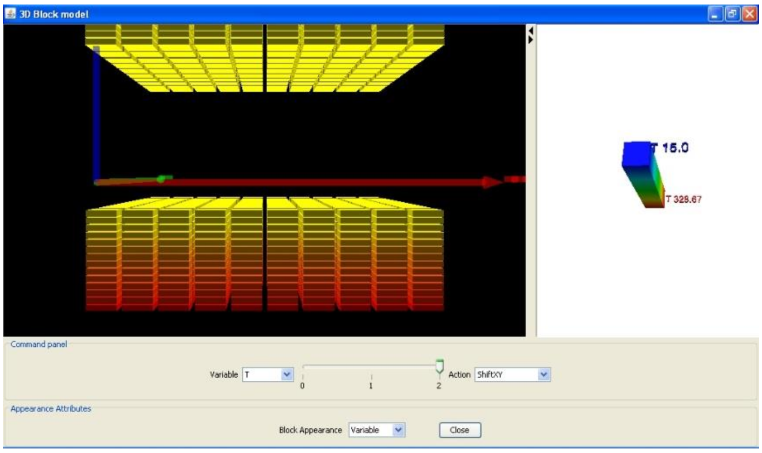


Fig. 22 - 3D modelFrame horizontally sectioned.

From the Action combo box, choose **ResetPositions** command to reassembly the 3D model Frame, and then select the command **Action->Get2Plot**. Move the mouse on the 3D frame plot and left click on a block to select it.

A Profiles dialog window is displayed, as showed in Fig. 23 and Fig. 24. One can select another variable to plot, acting on the "Variable" combo box, or change the time step by moving the time step slide bar. The "Domain" combo box allows to create a scatter plot of the chose variable, versus one of the Cartesian direction (x, y or z) or versus time.

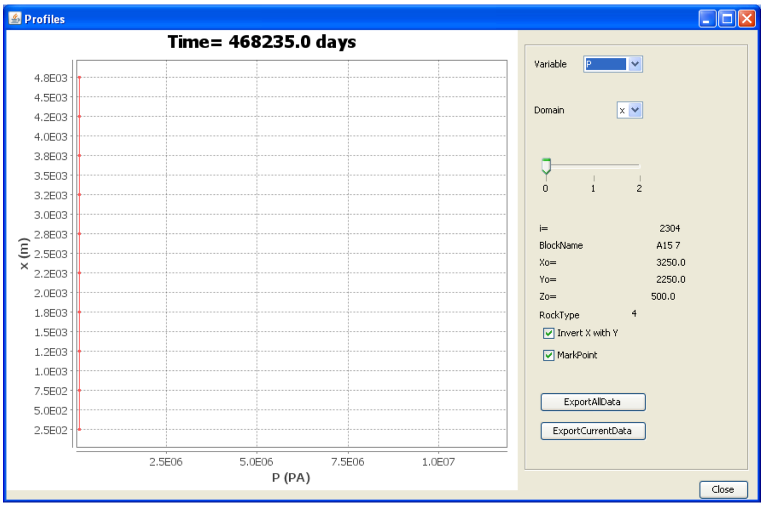


Fig. 23 – Graph of the pressure versus the x coordinate.

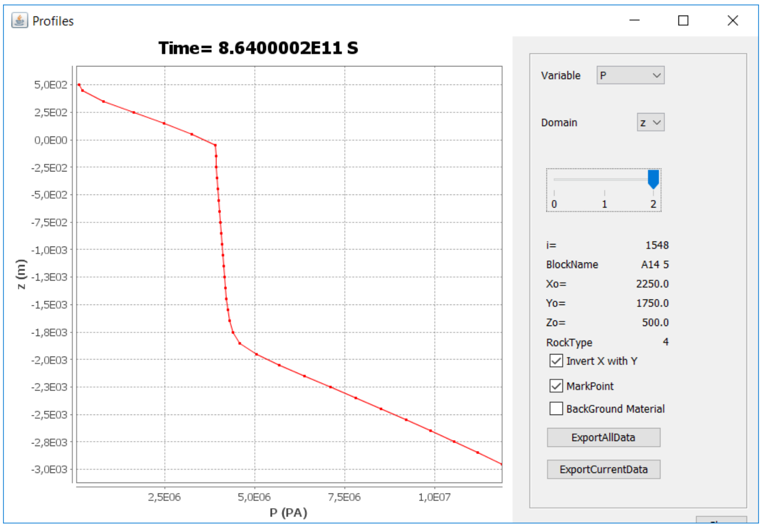


Fig. 24 – Graph of the pressure versus the z coordinate.

For structured grids, is possible to have the background of the plot coloured by the rocktype color scale. Click in option box **Background Material** to obtain the plot as shown in Fig. 25. In order to modify the rocktype scale, open the **Option** dialog window (accessing it from **Tools->Options->tab 3D Model object**).

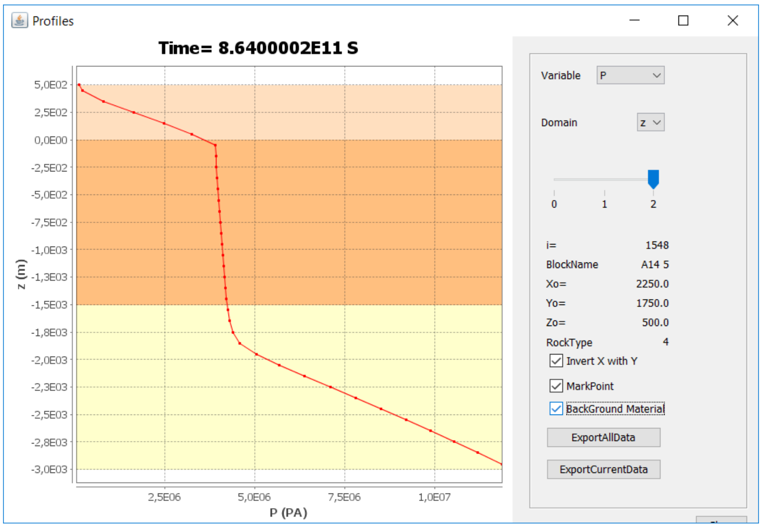


Fig. 25 – Graph of the pressure versus the z coordinate, with background coloring.

Close the **Profiles** dialog window by clicking the window icon  or the **Close** button.
Select the **Action->ExpandZ** command to explode the 3D Model Frame of the all layers (see Fig. 26).

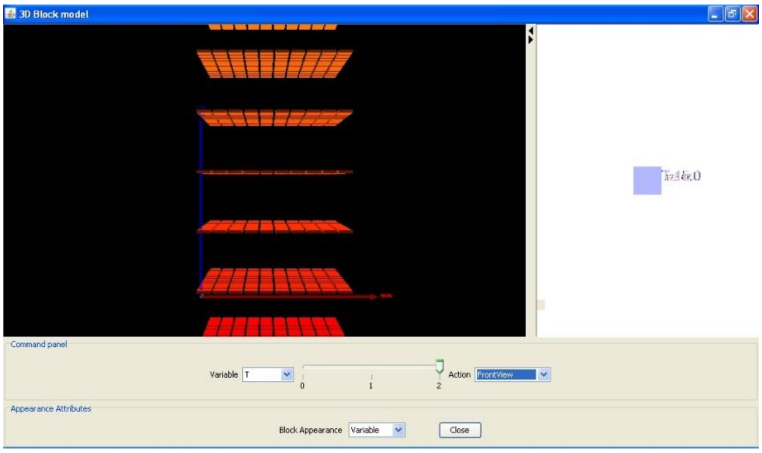


Fig. 26 - ExpandZ command result.

Select both **Action->ResetPositions** and then **Action->QuickInfo** commands. Then clicking a block and to obtain information about it reported at the bottom of the “3D Block model” dialog window (see Fig. 27).

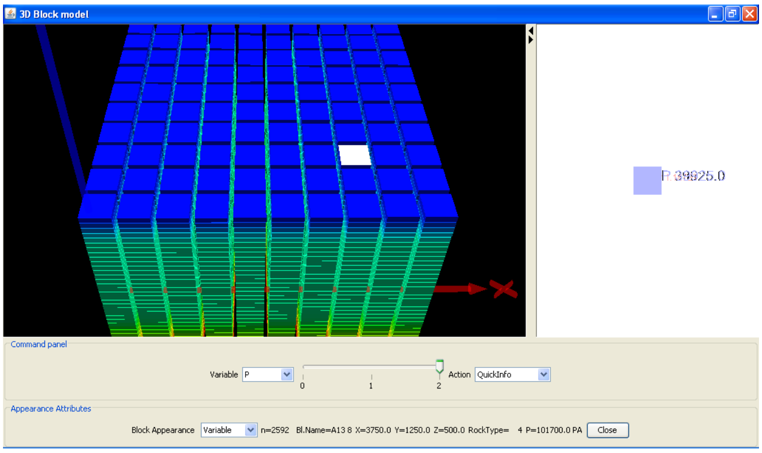


Fig. 27 - QuickInfo command result, information are printed in a panel at the bottom of the window.

Select the **Action->SetVOI** command to set the coordinates of a volume of interest. When the **SetVOI** dialog window is displayed. Select the "**Enable**" check box, click the **Default** button to set a default volume to be hidden (will be hidden a volume corresponding to half dimension on the three Cartesian direction) and click the **Apply Hide VOI** button. The 3D Block model is modified as shown in Fig. 28.

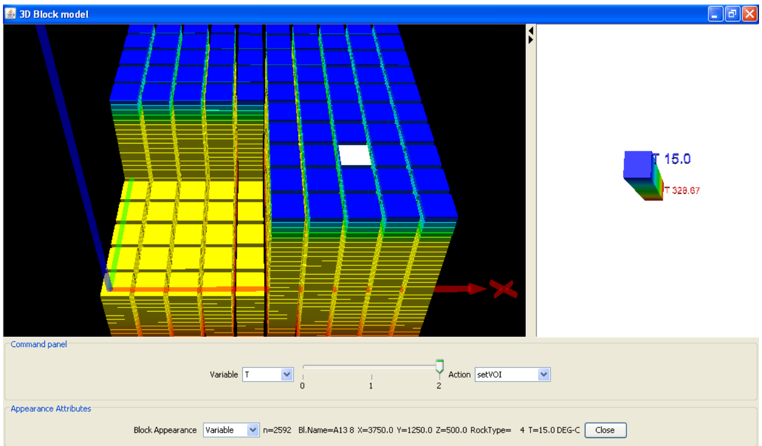


Fig. 28 - SetVOI function applied to the model.

From the **Block appearance** combo box, select **RockTypes.** The block scale colour is related to the rock type (as shown in Fig. 29).

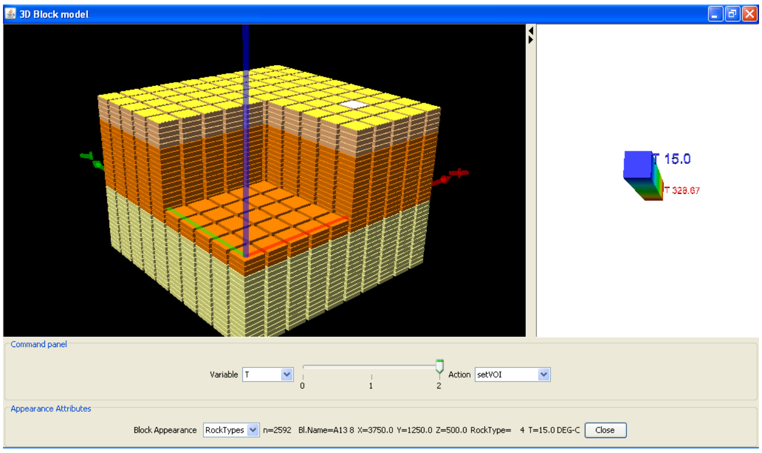


Fig. 29 - 3D Block model in material color scale.

Note that the color scale frame on the right is not update with rock type colors. The legend of rock type color is visible in a table present in the Option dialog window. From the main menu select **Tools->Option** to display the Option dialog window and click **3D Block model object** tab. To send to the printer the rock type color scale, click the **Print** button.

Close the **3D Block model** and the **Option** dialog by clicking the window icon  or the **Close** button.

6.2.4 3D Flow Vector visualization

This functionality gives a 3D vector representation of the heat or mass exchange among blocks. From main menu, click **View->3D Flow Vector** and the **3D Flow vector** dialog window is displayed as shown in Fig. 30.

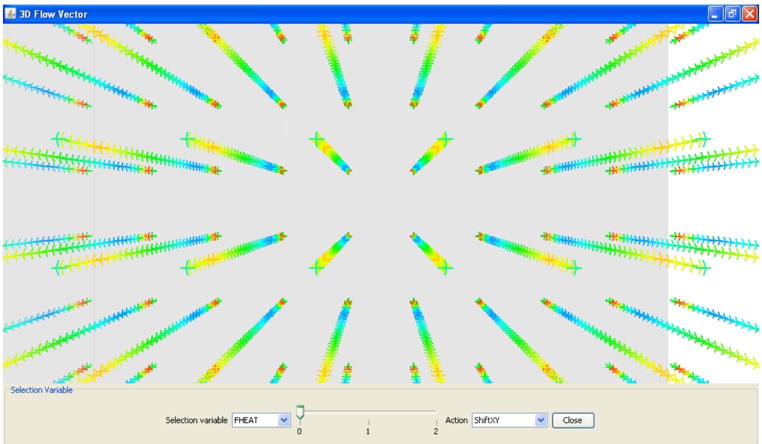


Fig. 30 - Vector representation of the heat flow.

From the **Selection variable** combo box select FHEAT. Move the slider to “**2**” and manage the model with usual CAD command.
From the **Action** combo box, select **Get 2D Plot** command, then click a vector on the 3D model and then the default **Vertical Flow Profiles** dialog window is displayed (see Fig. 31).

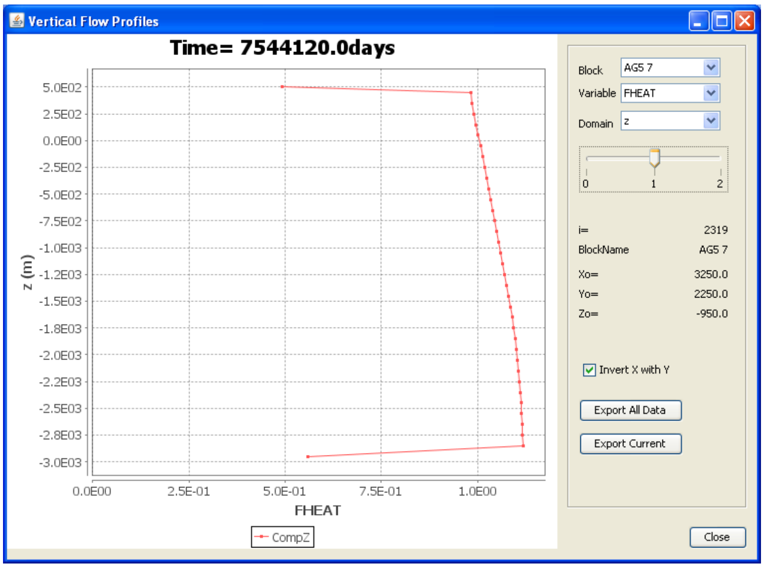


Fig. 31 - Vertical heat flow profile.

From **Block** combo box, is possible to change the block to use.
From **Variable** combo box, select a different flow variable and from the **Domain** combo box choose **z** (versus **z** axis) or **t** (versus time)

Close the **Vertical Flow Profile** dialog by clicking the window icon or the **Close** button.

Close the **3D Flow vector** dialog by clicking the window icon or the **Close** button.

6.2.5 3D Iso Surface

From TOUGH2Viewer menu bar, click **View->3D IsoSurface** (that is the three-dimensional analogue of the isolines), and then a **3D IsoSurface** dialog window is displayed as shown in Fig. 32.

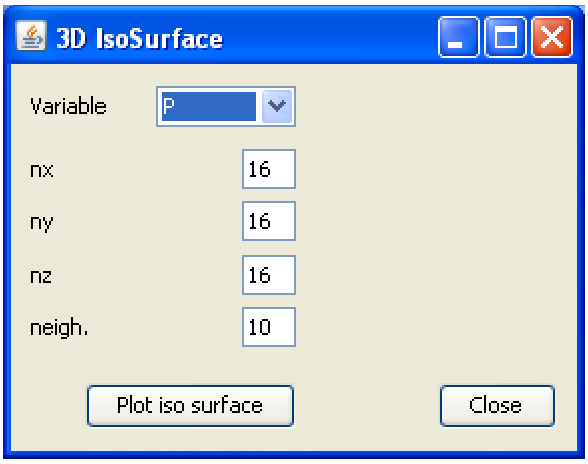


Fig. 32 - 3D IsoSurface dialog.

Click **Plot iso surface** button to obtain a **IsoSurface** dialog (see Fig. 33).

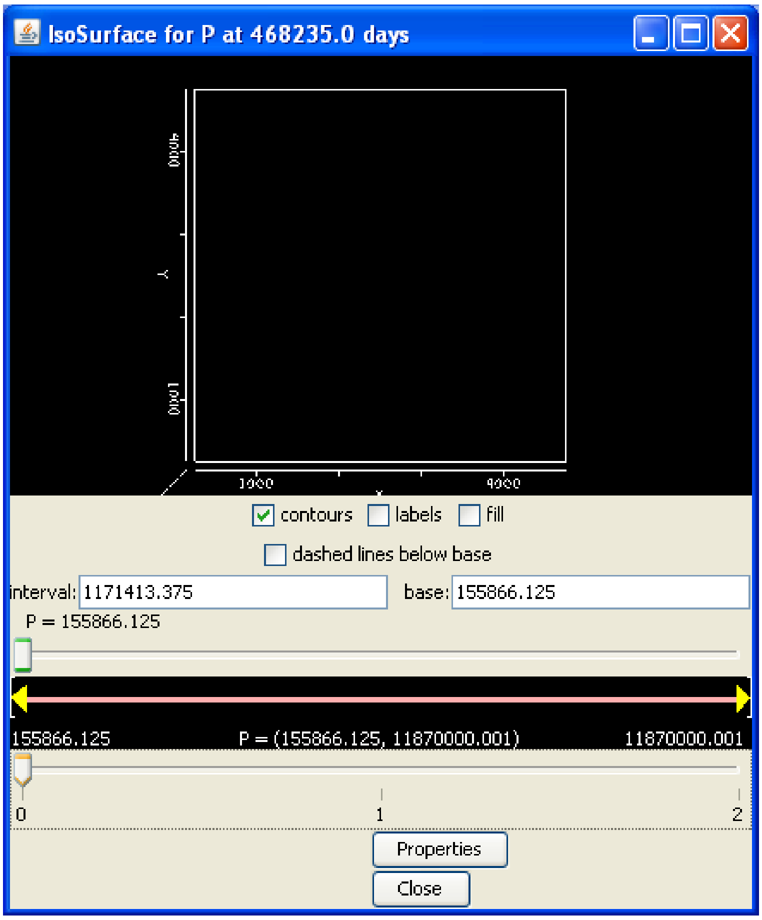


Fig. 33 - IsoSurface Dialog for pressure and time step 0.

In the **IsoSurface** dialog (the base is the minimum value of the selected variable), rotate the model from top view with usual CAD commands. Move the **slide bar** to set a iso value or move the time step slide bar and observe surface modification. An example is shown in Fig. 34.

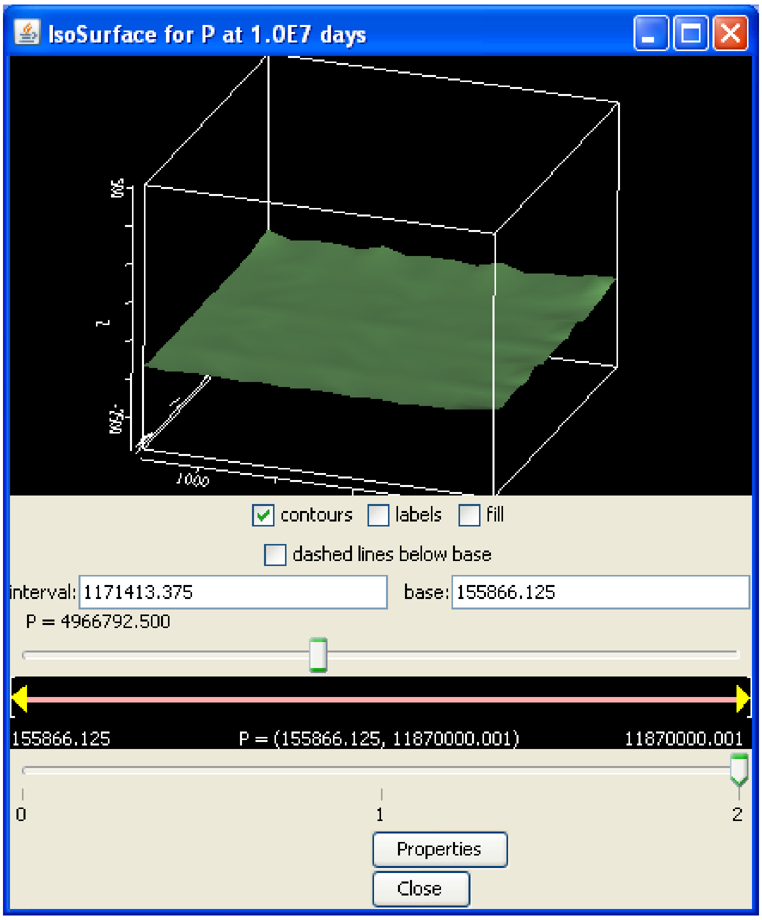


Fig. 34 - Dialog window for the pressure IsoSurface.

Close the **IsoSurface for...** dialog by clicking the window icon or the **Close** button.
Close the **3D Iso Surface** dialog by clicking the window icon or the **Close** button.

6.2.6 2D Contour Plot visualization

This functionality enables to plot the primary variables by means of contour lines.
From main menu, click View->2D Contour plot. A Contour Plot dialog is displayed. From the **Variable** combo box, choose **P** (Pressure), move the slide bar to **2**, select the “**XZ**” item from **Sectioning Plane** combo box. In the **y=** text box, type **500** (to specify the y coordinate of the sectioning plane). Click the **Plot** button and the **Contour Plot** dialog is updated as shown in Fig. 35.

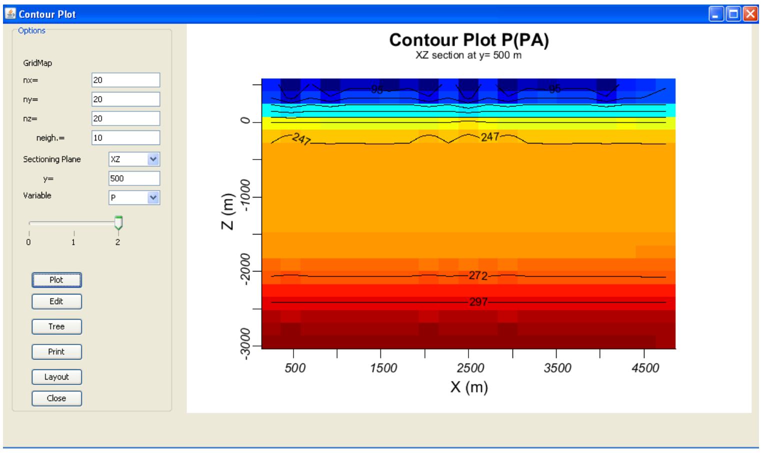


Fig. 35 - 2D Contour Plot for the Pressure (Pa).

Close the **Contour Plot** dialog by clicking the window icon  or the **Close** button.
To get basic statistic information about the simulated data, from main menu, click **Analysis->Statistics** menu item and then the “Statistics” dialog window is displayed (see pag. 16).
Exit form **TOUGH2Viewer** by clicking the window icon  or by **File->Exit** menu item.

6.3 Unstructured 3D grid model

This example is related to an unstructured grid model, built using a special GIS GRASS version (Berry et al, 2014) adapted to create locally refined grids.
Please visit http://software.dicam.unibo./tough2gis web pages for further detail.
The example folder contains the following files and folders:

- 3d_unstruct_03.out: simulation results
- in.dat: the input file for AMESH
- segmt.dat: output file of AMESH

Subfolder Maps:

- Subfolder base_ss: contains the 3D iso contour of the temperature:
  - base_ss.dbf
  - base_ss.prj
  - base_ss.qix
  - base_ss.shp
  - base_ss.shx
  - base_ss.xls
- Subfolder Raster
  - srtm_dem: gridded ASCII files of the digital elevation model of the topography.

Start **TOUGH2Viewer** (Windows OS) by double clicking the batch file **go.bat** then the main window of TOUGH2Viewer is displayed, as shown in Fig. 36.

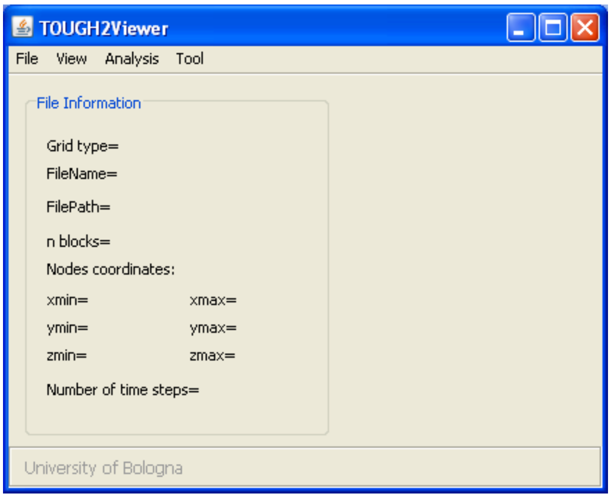


Fig. 36 - TOUGH2Viewer empty main window

6.3.1 Load Files

From main menu, click the **File->Open unstructured grid** menu item.
The **Open unstructured grid** dialog window will be displayed (Fig. 37).

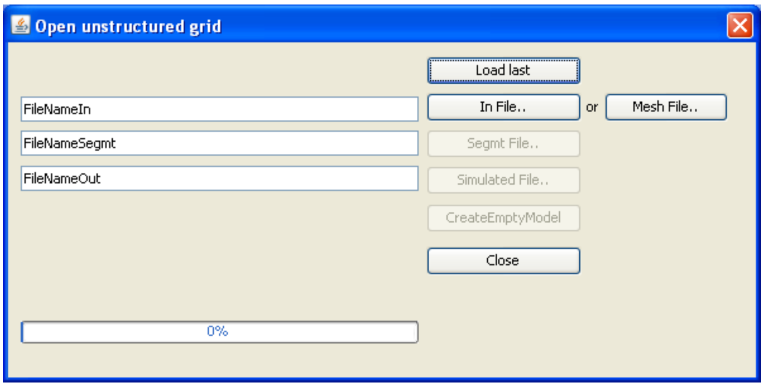


Fig. 37 - Open unstructured grid dialog window

Click the **In File** button to open the **Open In** dialog window and select the file called “in.dat” from the subdirectory “\T2Viewer\ExampleData\03_Unstructured_3D\”.
Click **Open** to start reading this file. Once finished the text box on the left side contains the full path of the file.
Click the **Segmt File** button to open the **Open Segmt** dialog window and select the “segmt.dat” file.
Click **Open** to start loading this file. When the loading is finished, the text box on the left side contains the full path of the file.
Click the **Simulated File** button to open the **Open out** dialog window and select the **“01_ss_dir_01.out”** file.
Click **Open** to start loading this file. The progress bar of the **Open unstructured grid** dialog shows the reading process. Close the **Open unstructured grid** dialog by clicking **Close** button to return to the main window.
The main window now shows some geometric information about the loaded model (see Fig. 38).

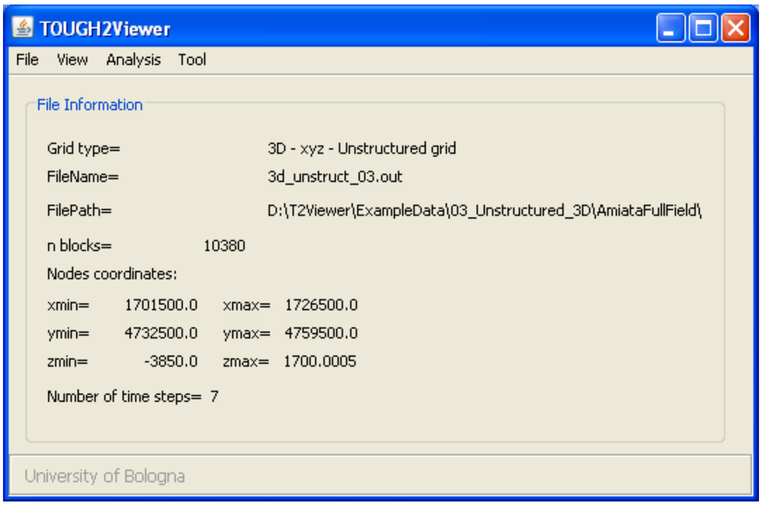


Fig. 38 - TOUGH2Viewer main window that summarizes some geometric data of the loaded numerical model.

In order to to load shapefiles to be included in the 3D Model Frame, from the main menu select the **File->Import shapefile** menu item and the **Import shapefile** dialog window is opened (Fig. 39).

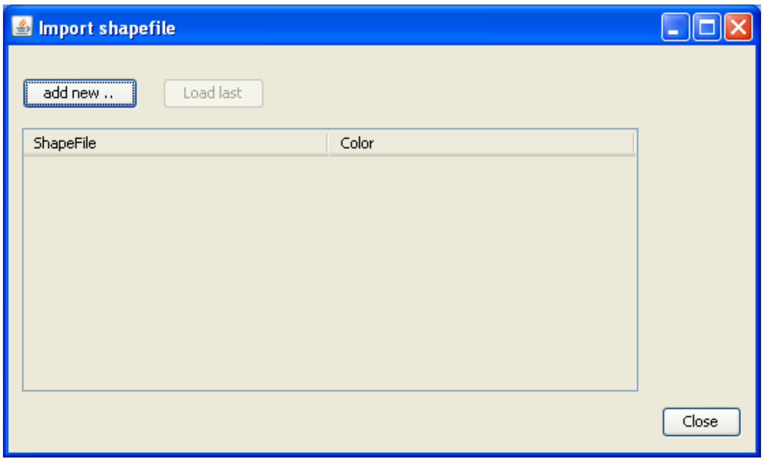


Fig. 39 - Import shapefile dialog window.

Click the **add new ...** button to open the **LoadShapeFile** dialog window (Fig. 40).

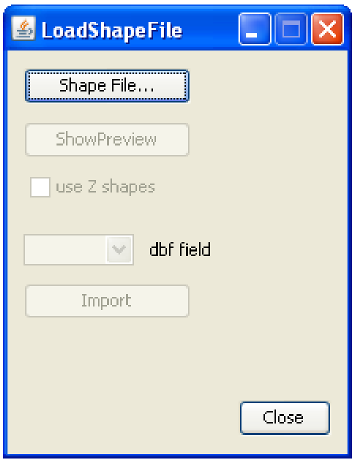


Fig. 40 - Dialog window to load shape files.

Click the **Shape File...** button to display the **Open** dialog window and navigate to the **base_ss** folder. In particular go to the "\ExampleData\03_unstructured_3D\Maps\base_ss" folder and select the "**base_ss.shp**" file (Fig. 41).

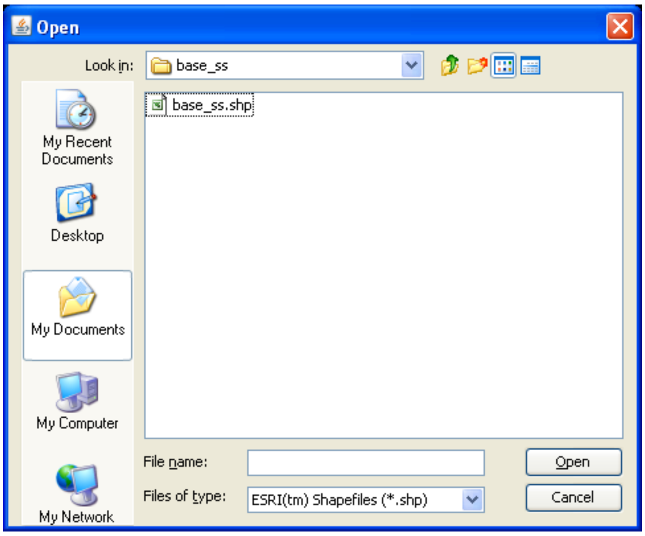


Fig. 41 - Dialog window to select the file to be loaded.

To obtain a 2D preview of the base_ss.shp file, click **Show Preview** button of the Load Shape File box and the **Shape file preview** window is displayed (Fig. 42).

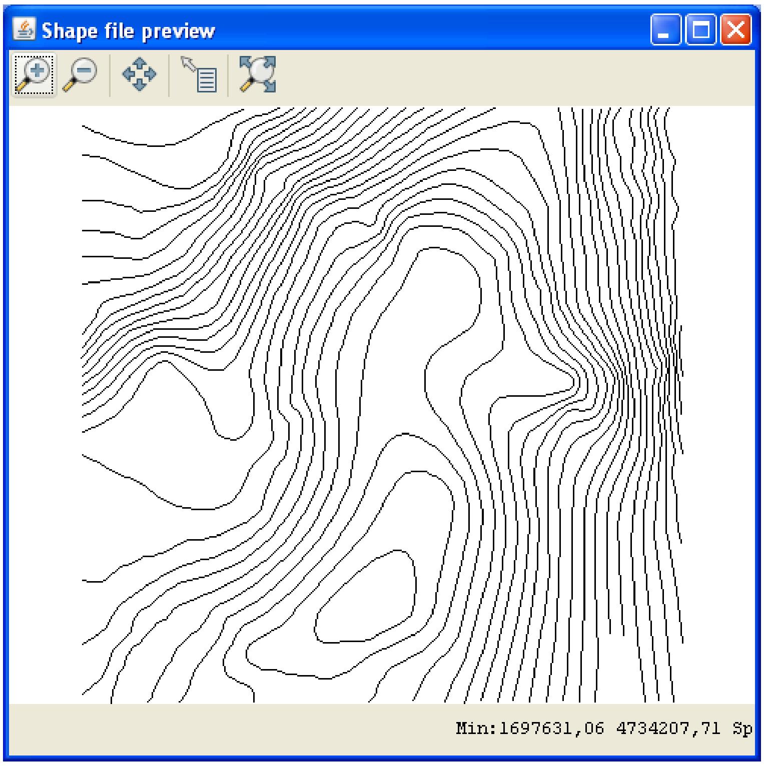
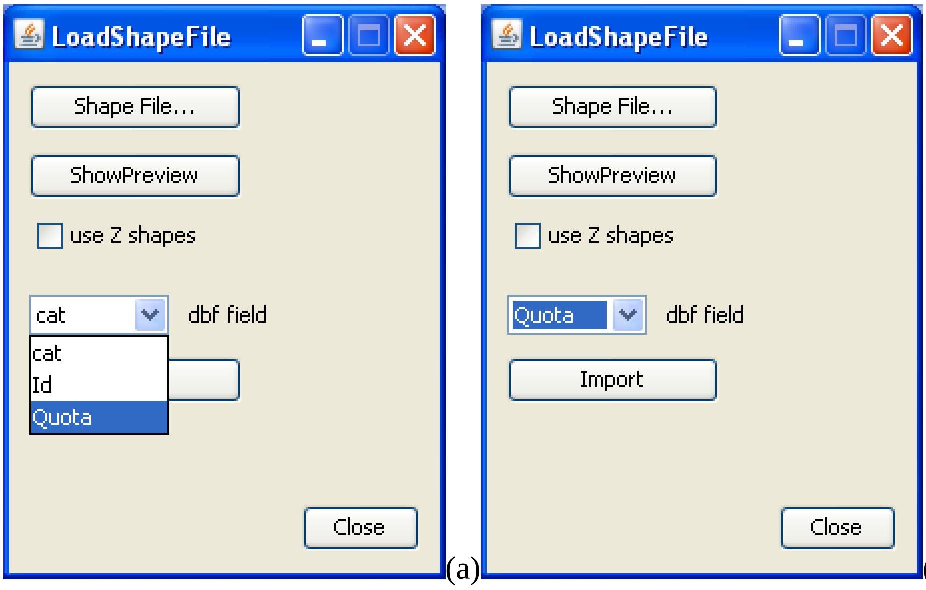


Fig. 42 - 2D Preview of the base_ss.shp file

Close the **Shape file preview** window by clicking the window icon . From the **LoadShapeFile** dialog window, select "**Quota**" item in the **dbf field** combo box (to load from a dbf file the elevation quote of the polylines object) (Fig. 43a) and click **Import** button (Fig. 43b).



(a) (b)

Fig. 43a, b - LoadShpFile Dialog window

The **Import shapefile** window now will be as shown in Fig. 44. The maximum number of loaded maps depends on the quantity of free RAM memory available.

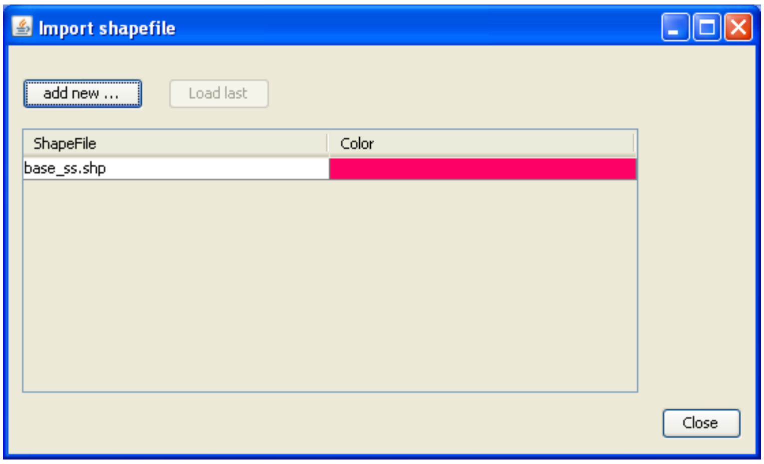


Fig. 44 - Import shapefile window. The table resume imported shape files.

By double clicking the **color** bar near the file name, the **Choose a color...** window dialog window is displayed to change the color of polylines (Fig. 45). Choose a color to plot the curve of the shape map and click **OK**, then close the **LoadShapeFile** window by clicking the window icon  or the **Close** button.

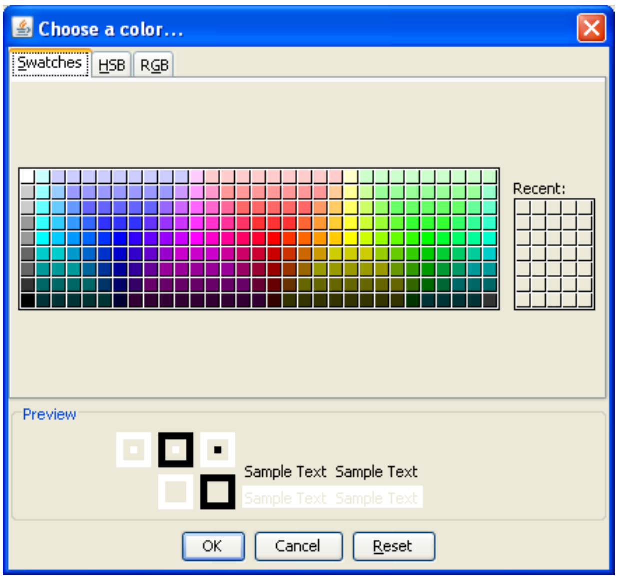


Fig. 45 – "Choose a color …" dialog window window.

Close the **Import shapefile** dialog window by clicking the window icon  or the **Close** button.

6.3.2 3D Block model view

From TOUGH2Viewer menu bar, click **View -> 3D block model** menu item and the **3D Block model** dialog window display, as shown in Fig. 46.

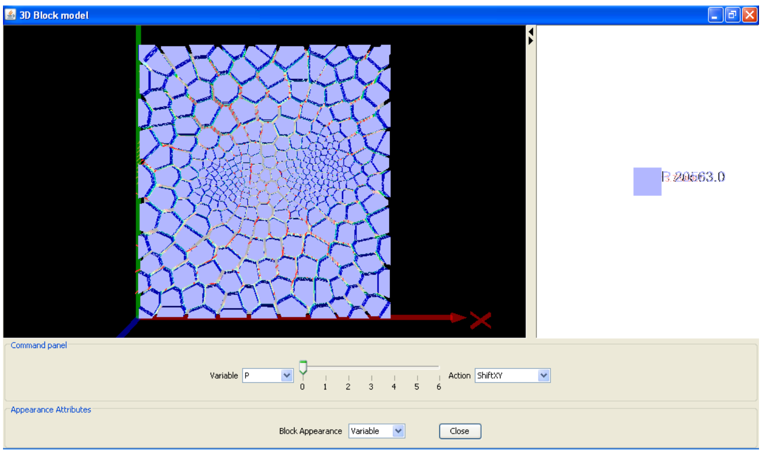


Fig. 46 - Top view of the 3D Block model.

The isolines and surface representing the base_ss.shp loaded in the previous step now are inside the model, and can be showed.
One can rotate, zoom, pan the model etc. using the mouse. To change variable to be showed, select it from **Variable** combo box. To change the time step, move the slide bar and automatically the plot will be update.
From the **Action** combo box, select **SetVOI.** A **SetVOI** dialog window is displayed. In the panel **Hide VOI,** check the **Enable** checkbox, click both **Default** and **Apply Hide VOI** buttons (Fig. 47).

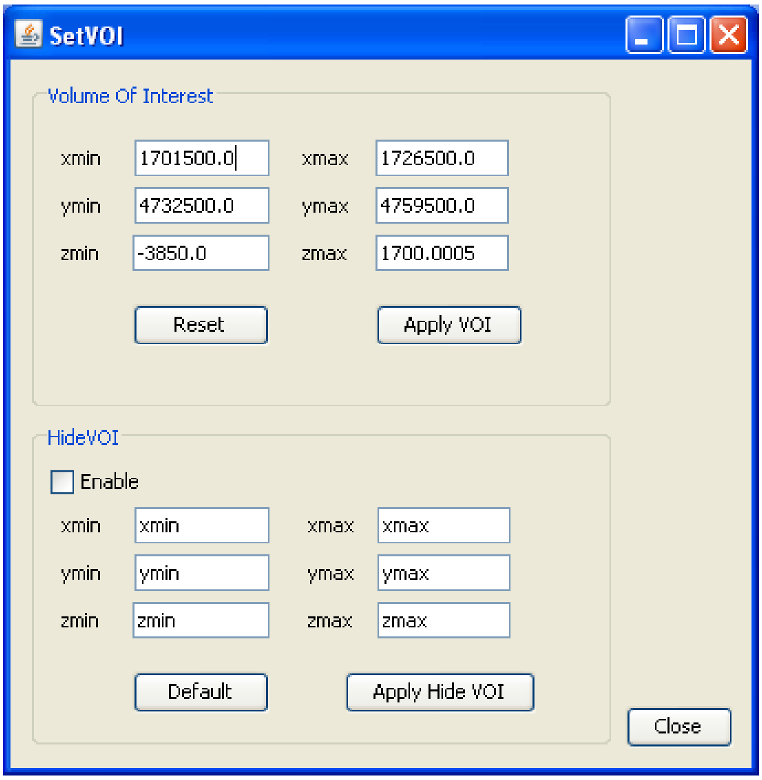
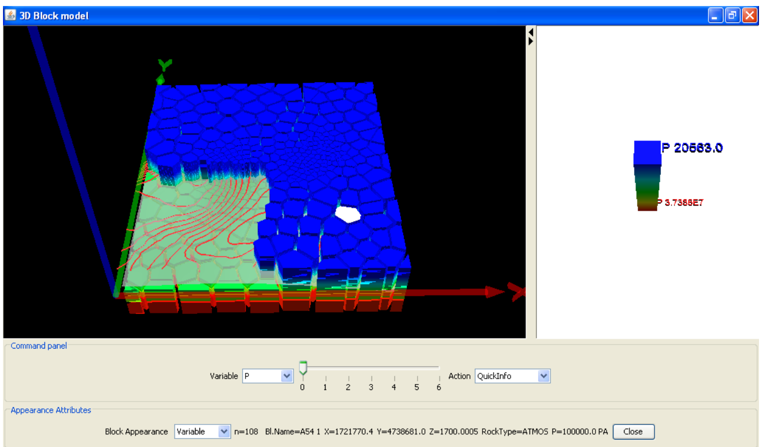
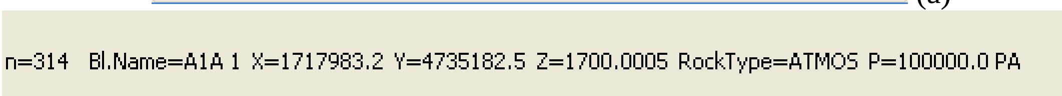


Fig. 47 - SetVOI dialog window.

Close the **SetVOI** dialog by clicking the window icon  or the **Close** button.
Is possible to rotate the view and minimize/enlarge the 3D frame by clicking the up black arrows (right-oriented arrow to minimize, left-oriented arrow to enlarge). The view of the model is now as shown in Fig. 39.
To obtain block information, choose **Action->QuickInfo**, then click a block on the 3D frame. At the bottom of the dialog window, a set of information about the selected block will be displayed (Fig. 39, (a)). These information will be automatically update when another variable or time step is selected (Fig. 48, (b)).



(a)



(b)

Fig. 48 - (a) 3D model partially showed and block selected (white block); (b) information related to the selected block.

From the **Action** combo box, select **FindBlocks** and the **Find Blocks** dialog window is displayed (Fig. 49). From the combo box, select a specific block (by the TOUGH2 name) and automatically in the **Block information** panel some information about it are showed and in the 3D model frame it will be highlighted.
To unselect the block, click **unselect** button. Close the **Find Blocks** dialog by clicking the window icon  or the **Close** button.

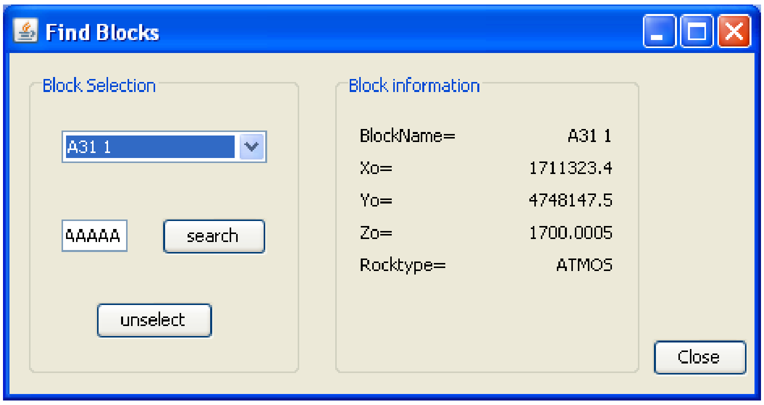


Fig. 49 - Find Block dialog window.

From the **Action** combo box, select **SnapShot** to display a **SnapShot Preview** dialog window that contains an image of the three-dimensional model (Fig. 50). With the **File->Save as...** menu item is possible to save the image (in png format) or send it directly to the printer by **File->print** menu item. Close the **SnapShot** dialog window by clicking the window icon  or via the **File->Exit** menu item.

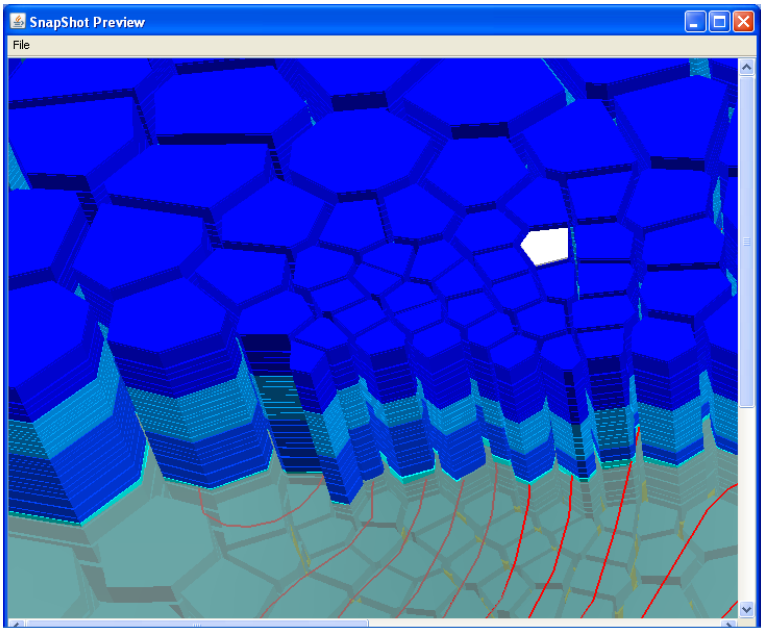


Fig. 50 - Snapshot Preview window dialog window.

To repaint each block in material scale color (Fig. 51), from the **Action** combo box of the **3D Block model** dialog window, execute the following commands:

- **ResetPositions**;
- **FrontView**;
- **SetVOI** and in the setVOI dialog window uncheck **Enable** check box and close setVOI dialog window;

From **Block Appearance** of the **3D Block model** dialog window**,** select **RockType**.

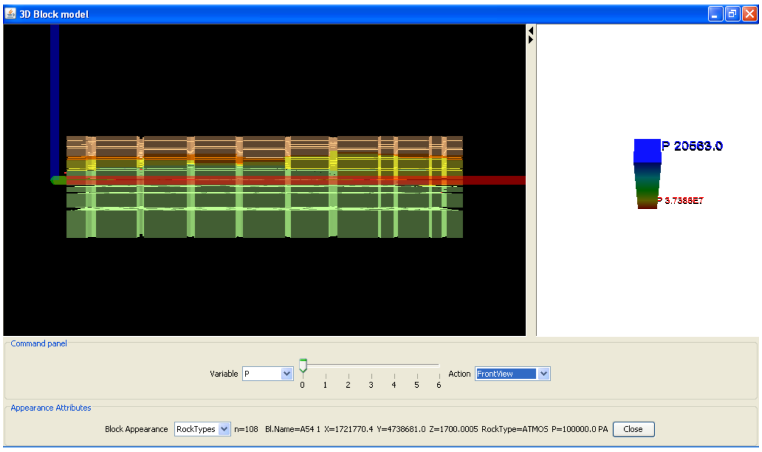


Fig. 51 - 3D Block Model front view. Material color scale.

From main menu, choose **Tools-> Options,** then click **3D Block model object** tab to open the Options dialog window (Fig. 52).

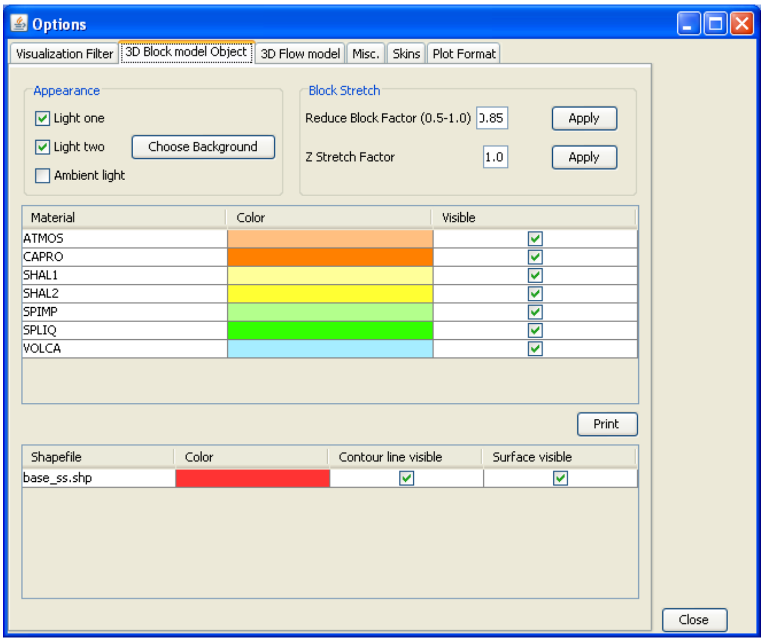


Fig. 52 - Options dialog window.

To obtain the view showed in Fig. 45, in the “Material” table of **3D Block model Object** tab, uncheck the check box of column **Visible** for:

ATMOS, CAPRO, SHAL1, SHAL2, VOLCA as show in Fig. 53. Only the materials SPIMP and SPLIQ are kept visible.

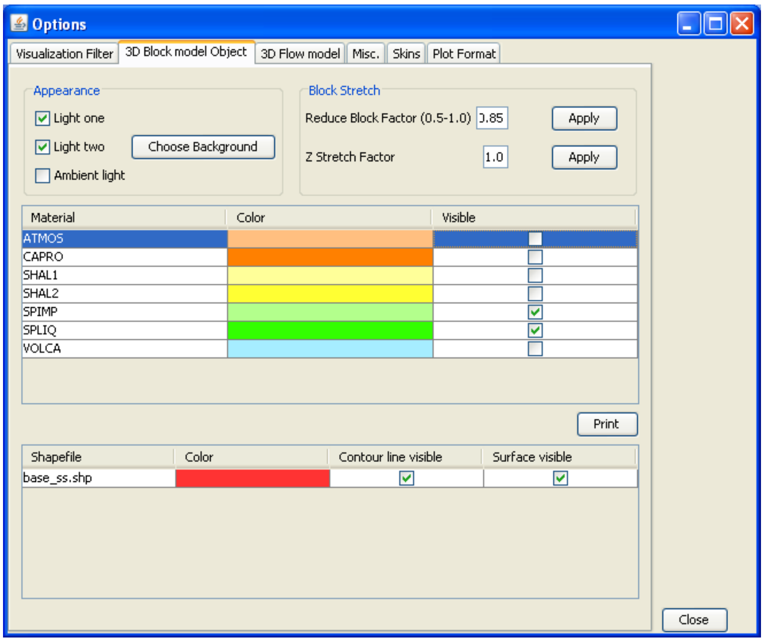


Fig. 53 - Options dialog with modified parameters.

To set not visible the contour line or the surface of shapefiles, uncheck the respective check box in the **3D Block model Object** tab. Close the Options windows using the window icon , or the **Close** button. **The 3D Block model** view now will appear as showed in Fig. 54.

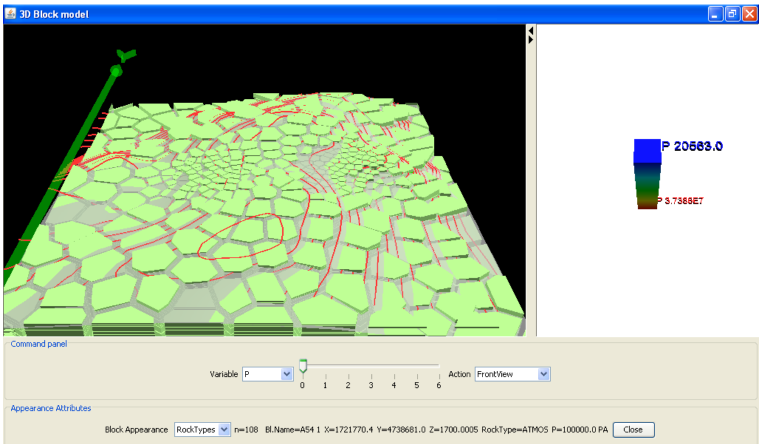


Fig. 54 - 3D Block model.

In the **Options** dialog window, **3D Block model Object** tab, restore ATMOS, CAPRO, SHAL1, SHAL2, VOLCA again visible.
In the **3D Block model**, from the **Action** combo box select **Get2DPlot** command. Click a block in the 3D frame and a **Profiles** dialog window is displayed.
From the slider menu select the time step “**6**”. The graph will be update as shown in Fig. 55.

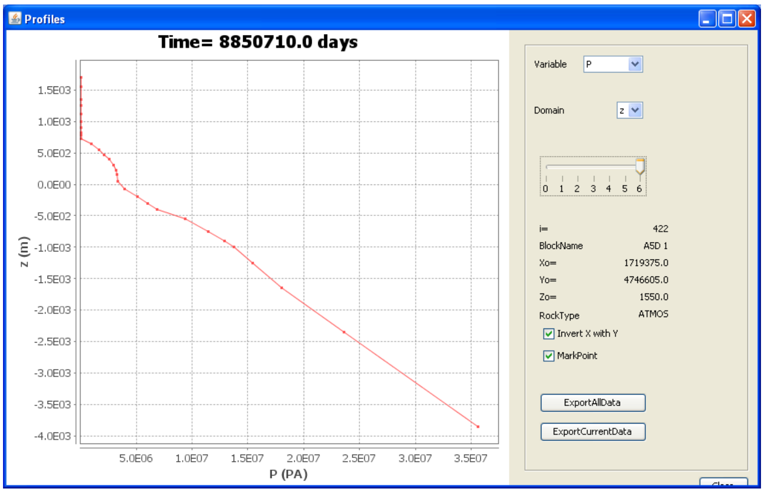


Fig. 55 - Profiles dialog which shows the pressure values along z axis.

Close the **Profiles** dialog window by clicking the  windows icon or the **Close** button.
In the **Options** dialog window, click on the **Visualization Filter** tab, select the **T** variable. In the **min** text box, type “270.0” and click the **setNewValue** button, and select the **HideOutOfRange** check box. In this way only blocks with temperature value between 270.0 and 291.01 °C are visible, note that also the colour scale, on the right frame, has changed as shown in Fig. 56.
In the **3D Block model** window, select **T** from **Variable** combo box and move the slider at position “6-th” (last time step present in this simulation output file).

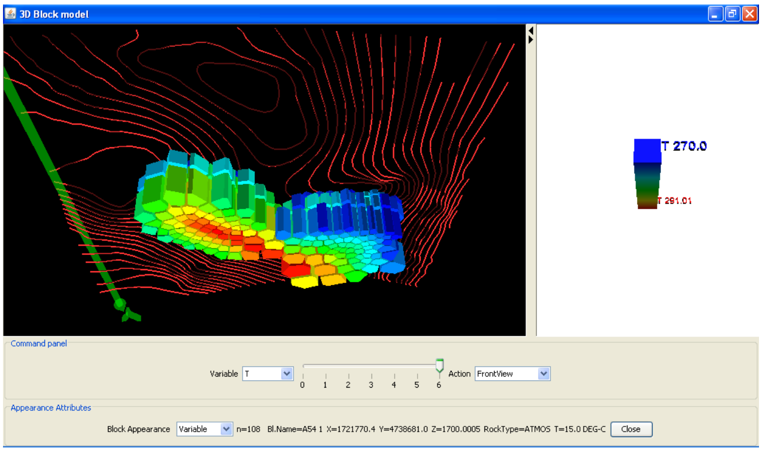


Fig. 56 - Bottom filtered view of the 3D model with modified color scale.

Close the **3D block Model** by clicking the  windows icon or the **Close** button.

6.3.3 3D Flow Vector visualization

This functionality gives a 3D vector representation of the heat or mass exchange among blocks. Select the **View->3D Flow Vector** menu item. A **3D Flow Vector** window opens, showing the default variable FHEAT, as shown in Fig. 57.

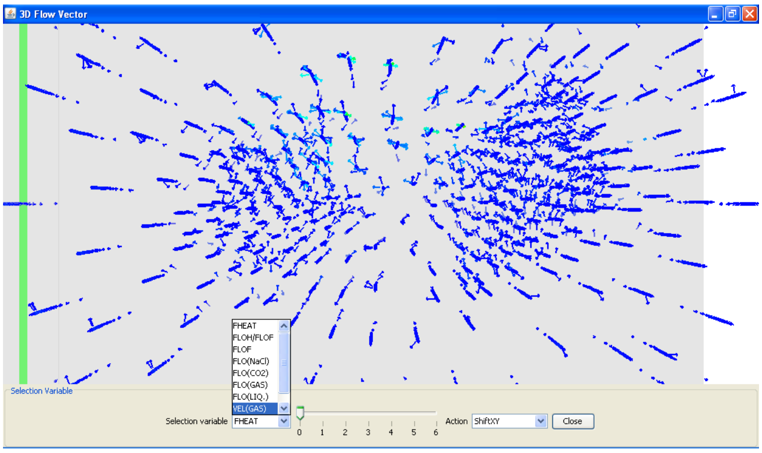


Fig. 57 - 3D Flow vector default viewing.

From the Variable combo box, select **Vel(GAS)** item and move the slider to **1**. With the usual CAD command zoom and rotate the model in the desiderated position (Fig. 58).

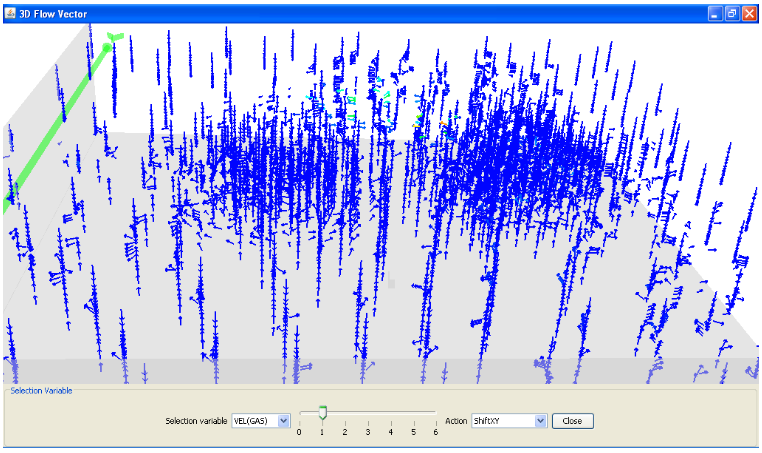


Fig. 58 - 3D Flow model at the time step 1 for the variable VEL(GAS).

Close the **Flow profile** window by clicking the  windows icon or the **Close** button.

Close the **3D Flow vector** window by clicking the  windows icon or the **Close** button.

6.3.4 3D Iso Surface

Select the **View->3D Iso Surface** menu item.

Open a **3D Iso Surface** dialog (Fig. 59) and from the **Variable** combo box, select **T** (default is P)**.** Leave other parameters as default and click **Plot iso surface** button. Wait few second to complete the interpolation process, and then a new **IsoSurface** window opens to plot the selected variable.

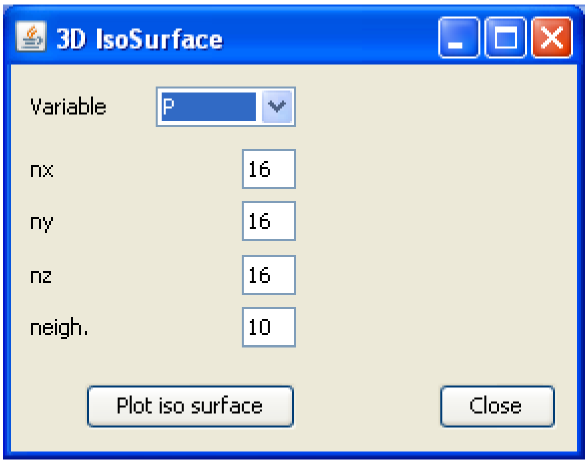


Fig. 59 - 3D IsoSurface dialog window.

In this window, move the first continuous **slider** to 270 °C. The plot windows will appear as like as shown in Fig 52. To see evolution of isosurface by time steps, move the time step slider under continuous slider. Click the **Properties** button to change colour background, to switch from parallel to perspective view, to change axis colour or hide the thin box.

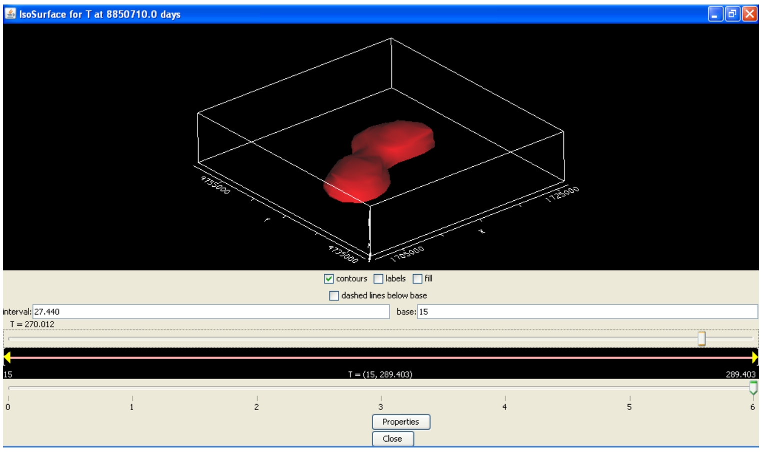


Fig. 60 - Isosurface for T=270°C, time step “6”.

Close the **IsoSurface for T** … and the **3D Isosurface** window by clicking the  windows icon or the **Close** button.

6.3.5 2D Contour map

Select the **View->2D Contour map** menu item and a **2D Contour plot** window will open.
In the **2D Contour plot** window, set the **Sectioning plane** combo box to **XZ**, and type “**4745000**” (vertical XZ section at y=4745000 m), move the slider at position “6-th” (last time step present in this simulation output file) and click the **Plot** button. A **Contour Plot** window will be displayed as shown in Fig. 61.

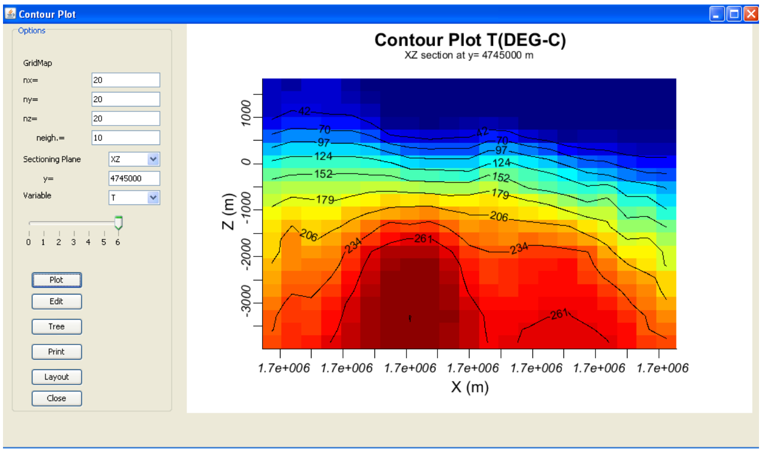


Fig. 61 – Temperature isolines contour map.

After having adjusted the parameter of plotting, click the **Plot** button to update the graph. Close the **2D Contour plot** window by clicking the  windows icon or the **Close** button.

6.3.6 Statistics

Click the **Analysis-> Statistics** menu item. The **Statistics** dialog window is displayed (see pag. 16.). Exit form **TOUGH2Viewer** by clicking the window icon  or by **File->Exit** menu item.

6.4 Voronoi 3D grid model

This example is related to a Voronoi 3D unstructured grid model, built using VORO2MESH (Bonduà et al., 2017) to create full Voronoi 3D grid. This grid is also a Central Voronoi Tessellation (CVT). The example folder contains the following files:

- MESH: the classical TOUGH MESH
- tough2viewer.dat: the geometry file, as generated by VORO2MESH;
- voronoi3D.out: the simulation output results of TOUGH simulation run.

Start **TOUGH2Viewer** (Windows OS) by double clicking the batch file **go.bat** then the main window of TOUGH2Viewer is displayed, as shown in Fig. 62.

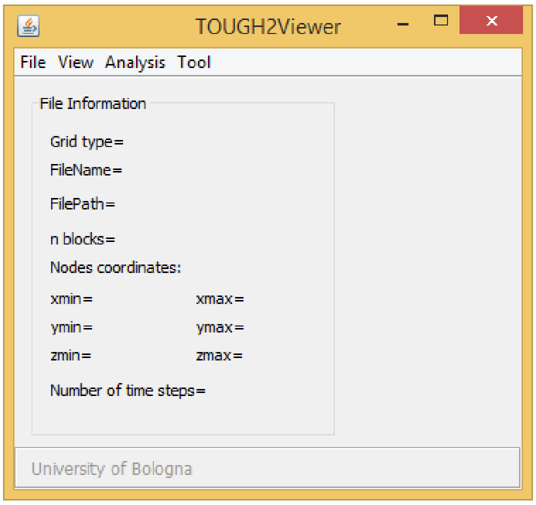


Fig. 62 - TOUGH2Viewer empty main window

6.4.1 Load Files

From main menu, click the **File->Open unstructured V++ grid** menu item.
The **Open unstructured V++ grid** dialog window will be displayed (Fig. 63).

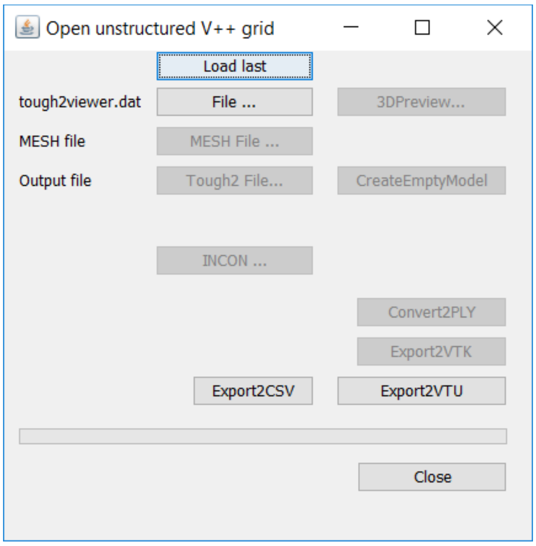


Fig. 63 – Input unstructured grid window

Click the **File** button to open the **tough2viewer.dat** dialog window and select the file called "tough2viewer.dat" from the subdirectory "\T2Viewer\ExampleData\04_voronoi_3D\".
Click **Open** to start reading this file. Once finished is possible to visualize a preview by clicking the **3DPreview** button on the right.
Click the **MESH File** button to open the **Open MESH** dialog window and select the "MESH" file in the same folder.
Click **Open** to start loading this file. Click the **Simulated File** button to open the **Open out** dialog window and select the **"voronoi.out"** file.
Click **Open** to start loading this file. The progress bar of the **Open unstructured V++ grid** dialog shows the reading process. Close the **Open unstructured V++ grid** dialog by clicking **Close** button to return to the main window.
The main window now shows some geometric information about the loaded model (see Fig. 38).

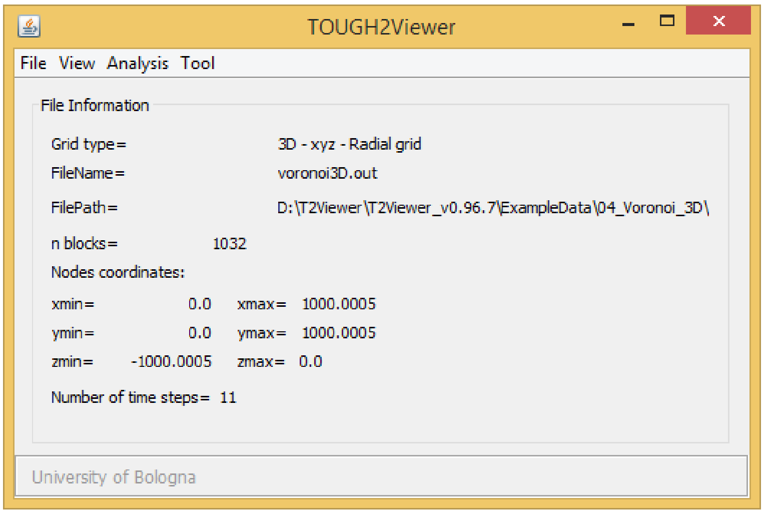


Fig. 64 – The main window

6.4.2 3D Block model view

From TOUGH2Viewer menu bar, click **View -> 3D block model** menu item and the **3D Block model** dialog window display, as shown in Fig. 65.

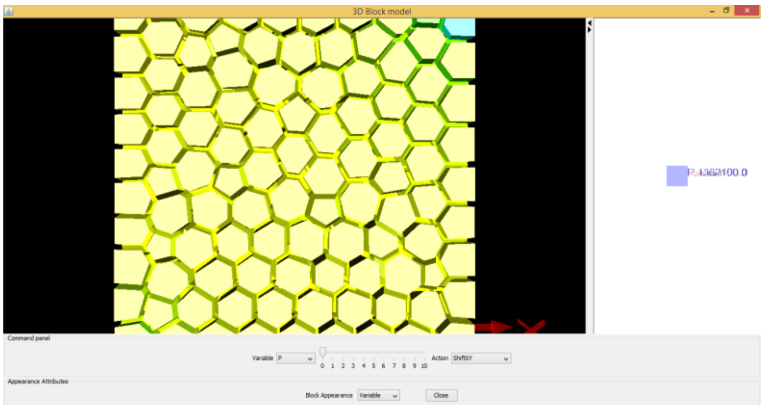


Fig. 65 - Top view of the 3D Block model.

Move the variable combo box to **SG** (Saturation of Gas). Rotate and change the time step progressively.
Visualization can appear as follow in Fig. 66.

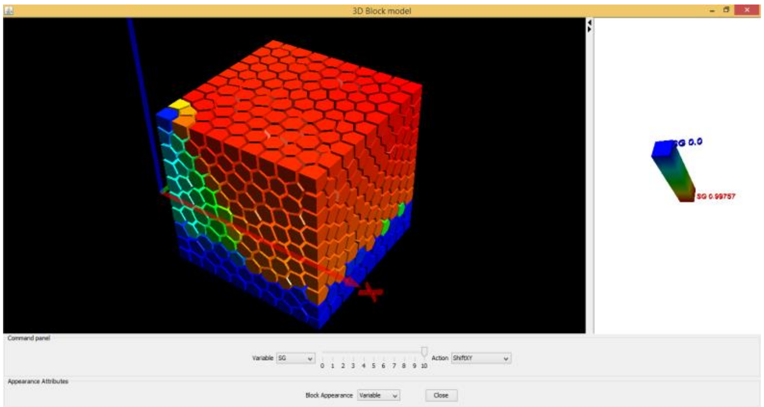


Fig. 66 – View of the SG thermodynamic variables (3D Block model).

Explore the model with the command as explained in the previous chapters.

6.4.3 Prepare GENER file

Select from the Action dropdown menu the **MultipleSelection** Command.
As shown in Fig. 67, select two top opposite blocks of the grid by clicking on it.

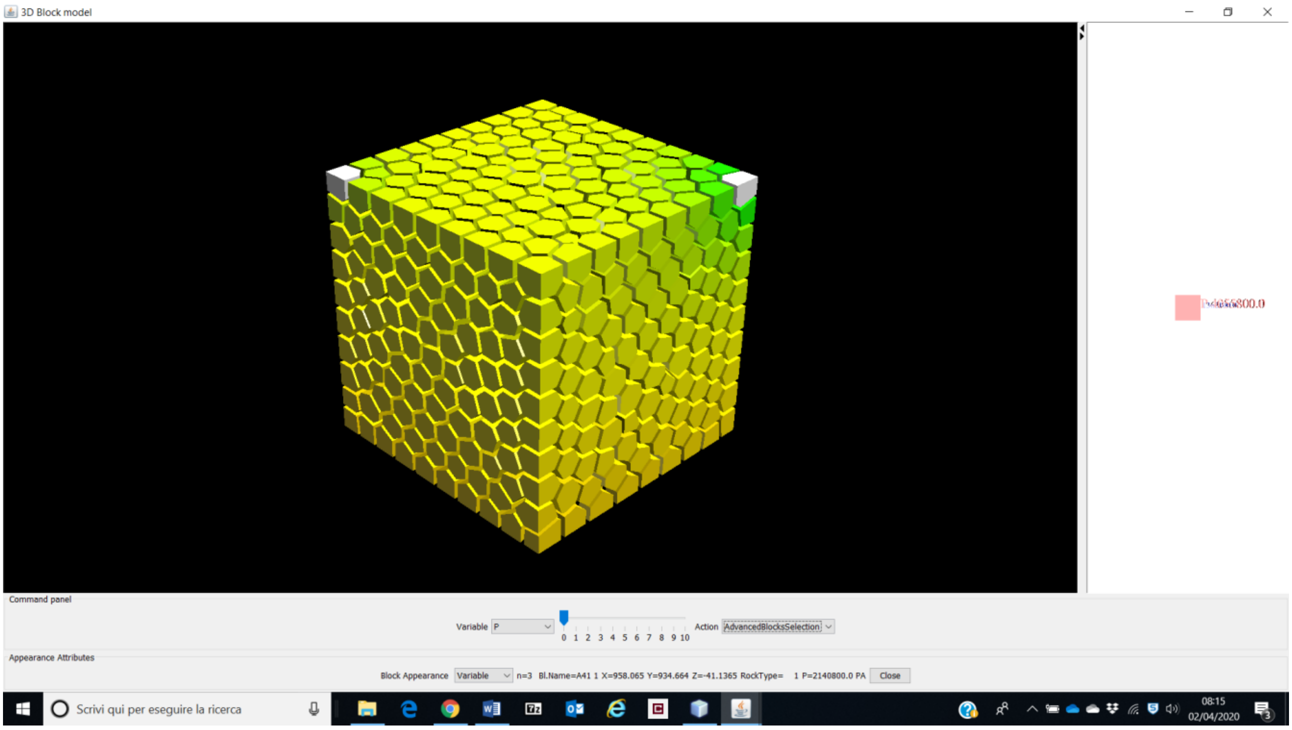


Fig. 67 – MultipleSelection command

Select from the Action dropdown menu the **AdvanceBlockSelection,** and then click on the button **Open modify Selection…**

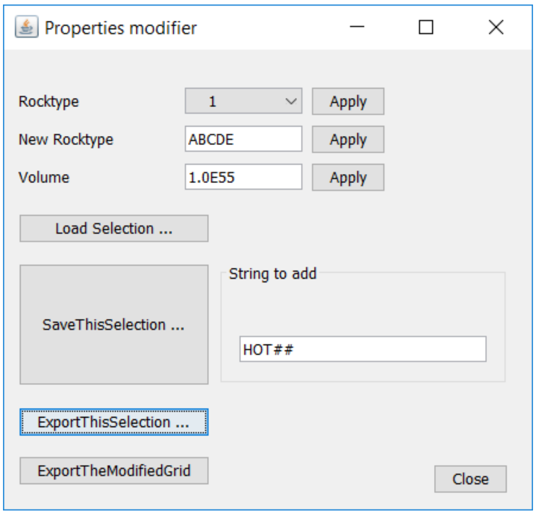


Fig. 68 – Properties modifier GUI

Click on **ExportThisSelection** and choose a filename, for example “generblocks”. An ASCII file will be created as shown in Fig. 69.

```
ELEME - MESH File for D:\D\T2Viewer\ExampleData\04_Voronoi_3D\twoblocks.dat genera
A11 1          *    1 8.855e+05                     5.669e+01 4.559e+01-4.190e+01
A41 1          *    1 8.687e+05                     9.581e+02 9.347e+02-4.114e+01

CONNE
```

Fig. 69 – The generated ASCII file

Remove all unwanted information and create your GENER file by using a text editor (Fig. 70).

```
GENER5
A11 1INJ 1                    1     MASS      3.75     8.0E4
A41 1PRO 1                    1     MASS     -3.75
```

Fig. 70 – The modified GENER file

INCON generation

Before creating a INCON file, is better to load a previous INCON or SAVE file as created, for example, by VORO2MESH or by a dummy run of TOUGH.
Load the INCON from file by the **Open Unstructured V++ grid…** dialog window.
Press the **Generate INCON** button. A new dialog window will be shown. From the variable selector select INCON_0 and then press the **add** button. Repeat the previous step up to the INCON_3 variable. Select from the INCON VARIABLES selector the INCON_0 variable and update the plot by pressing UpdatePlot. The Dialog will now have the aspect of Fig. 71.

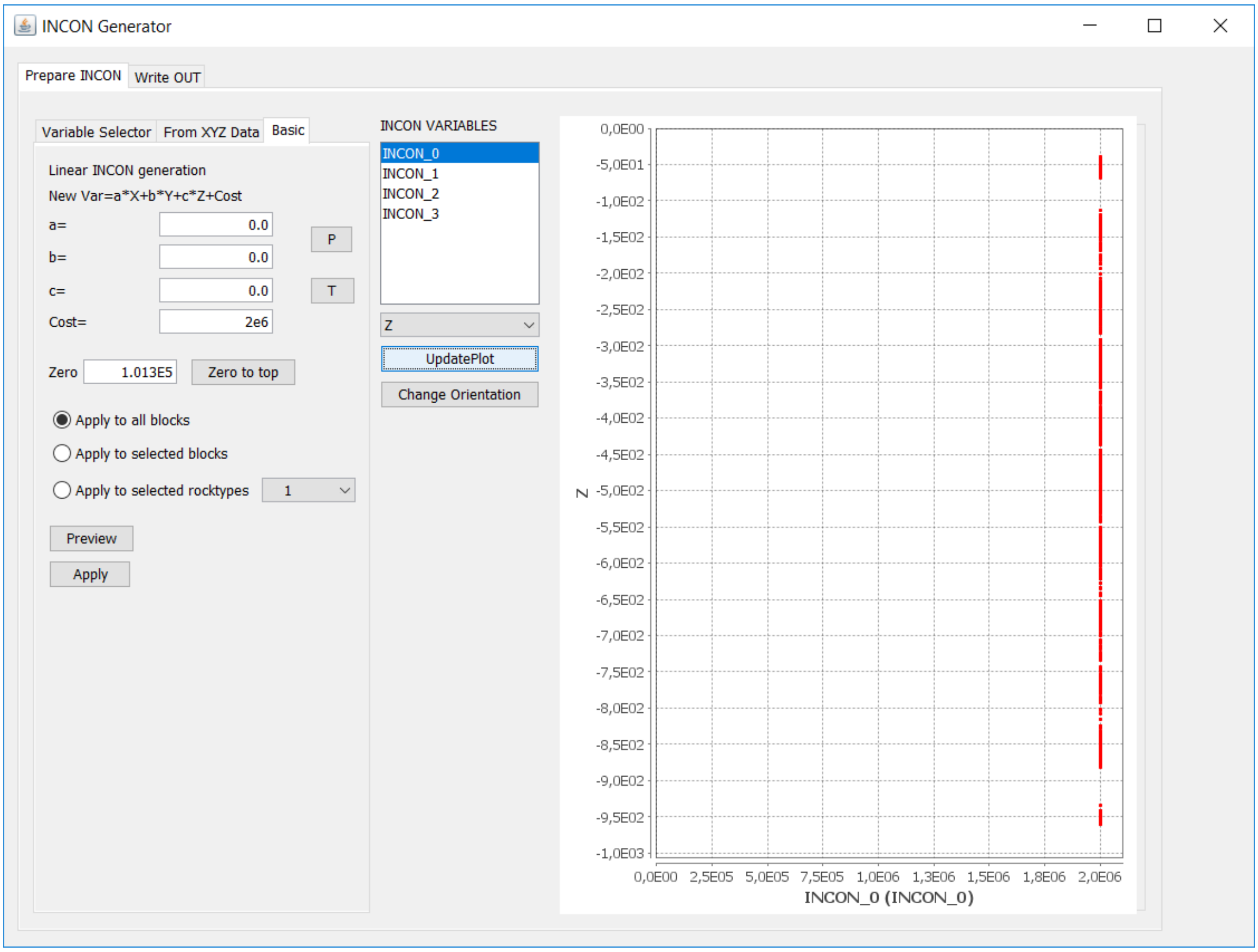


Fig. 71 – INCON generator dialog window

Set your value of INCON_0 (Pressure in this example) as 3.0E6 Pa and click on Preview button. In The new graph will be updated as shown in Fig. 72. Click on the **Apply** button to apply the changes.

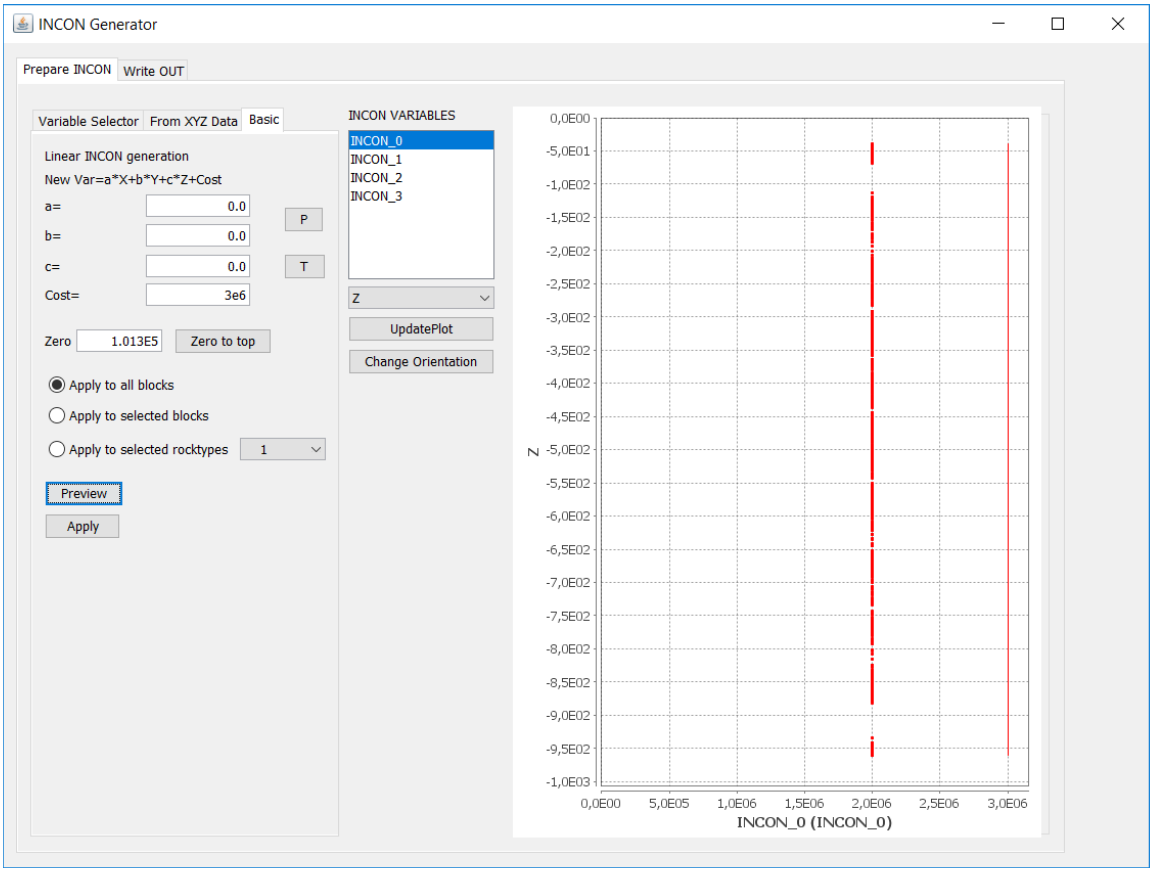


Fig. 72 – INCON generator dialog window preview

Click on **Apply**. Repeat all steps for the other INCON variables. When finished, move to the **Write Out tab**, and click on update preview, as show in Fig. 73.

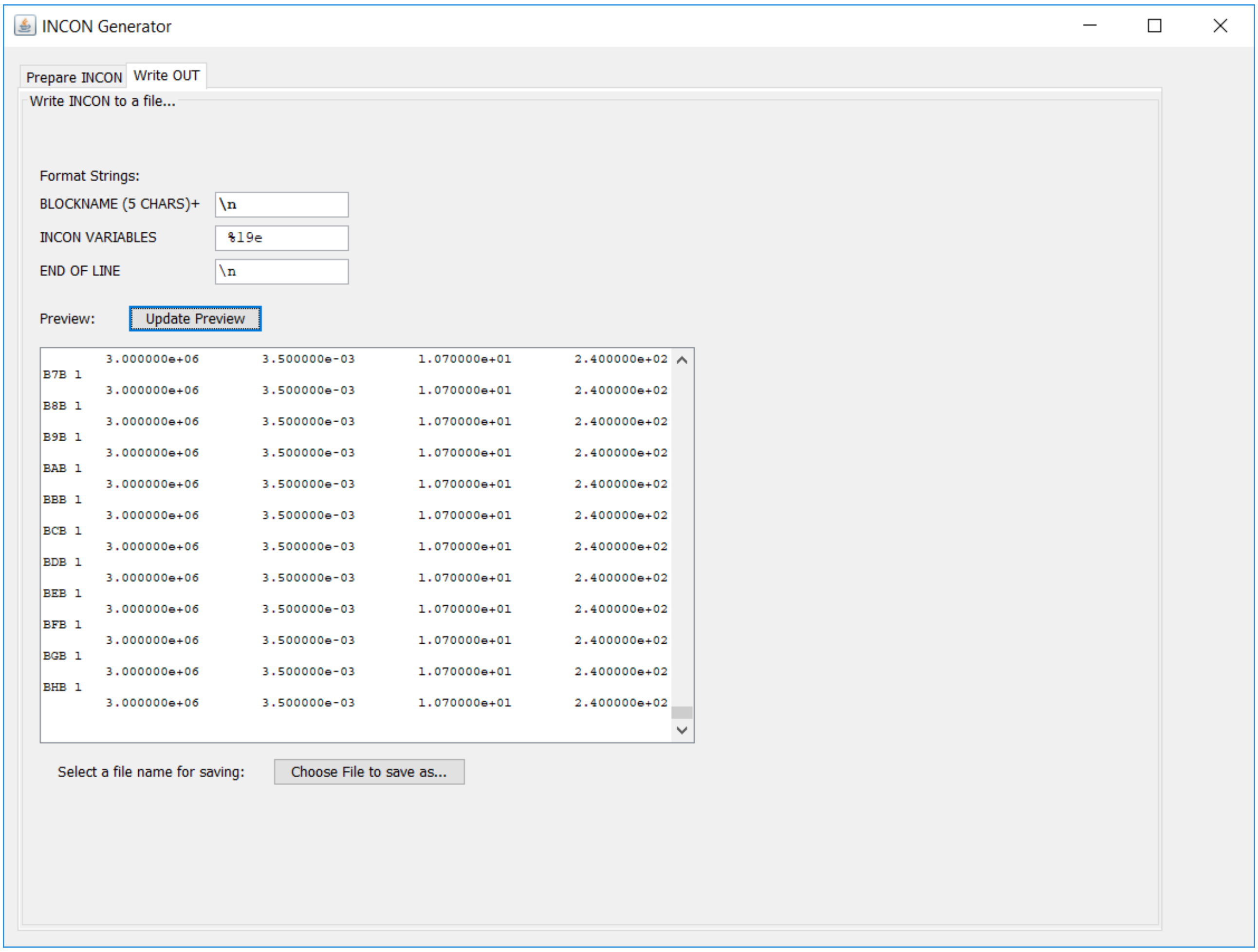


Fig. 73 – INCON generator dialog window –write out tab preview

To export the INCON data press on the **Choose File to save as…** and insert a name file for output.

6.4.4 Export data to Paraview

To export the model in a file format readable using Paraview, from the main menu, click the **File->Open unstructured V++ grid** menu item.

The **Open unstructured V++ grid** dialog window will be displayed (Fig. 63). Click the command button **Export2VTU.** A set of files (one for each time step) will be written in the same folder where data was read, named “out_[time_step_counter].vtu”. Open these files with Paraview to inspect simulation results.

6.5 The Amiata structured 3D grid case

In this example will be show how to handle material assignation of a pre-existing numerical "empty model" using raster surface.
In the folder \\ExampleData\ 05_Amiata3D\Structured_3D you will find the following files

01dem.dat
01dem.dat.ply
02fractured.dat
02fractured.dat.ply
03caprock_shal.dat
03caprock_shal.dat.ply
04shallow_res.dat
04shallow_res.dat.ply
05caprock_deep.dat
05caprock_deep.dat.ply
06bottom-5000.dat
06bottom-5000.dat.ply
voro.par
MESH
tough2viewer.dat

Each file represent the top surface of a different geological layer. In this presentation we just focus out attention on the rocktype assignation, without considering petrophysical properties of the geological layer, to be treated separately in the ROCKTYPE section of the TOUGH2 input file.
The coordinate of the surface and of the model are referred to geographica coordinates.
The limit of the domain of the numerical model are (m):
x_max=1726500.00
x_min=1701500.00
y_max=4759500.00
y_min=4732500.00
z_max=2000.00
z_min=-6000.00
The empty model builded using VORO2MESH, is composed by:

x_max	x_min	y_max	y_min	z_max	z_min
1726500	1701500	4759500	4732500	2000	-6000
Lx=	25000	Ly=	27000	Lz=	8000
nx	25	ny	27	nz	40
Tot. Blocks	27000				
Block dimensions					
dx=	1000	dy=	1000	dz=	200

Table 1 – Domain bound and grid information

We are using a finer discretization in the vertical direction.
The dimension of the blocks are: 1000x1000x200 m^3.
Open TOUGH2Viewer and form the file menu select Open unstructured V++ grid…
Select the tough2viewer.dat file and the MESH file from the path: \\ExampleData\05_Amiata3D\Structured_3D.

Close the dialog by selecting the close button.
From the file menu select Import Surface File…
Select, in the following order, the files:

- 01dem.dat
- 02fractured.dat
- 03caprock_shal.dat
- 04shallow_res.dat
- 05caprock_deep.dat
- 06bottom-5000.dat

After surfaces importation procedure, the **Import Shapefile** dialog window will be as shown in Fig. 74.

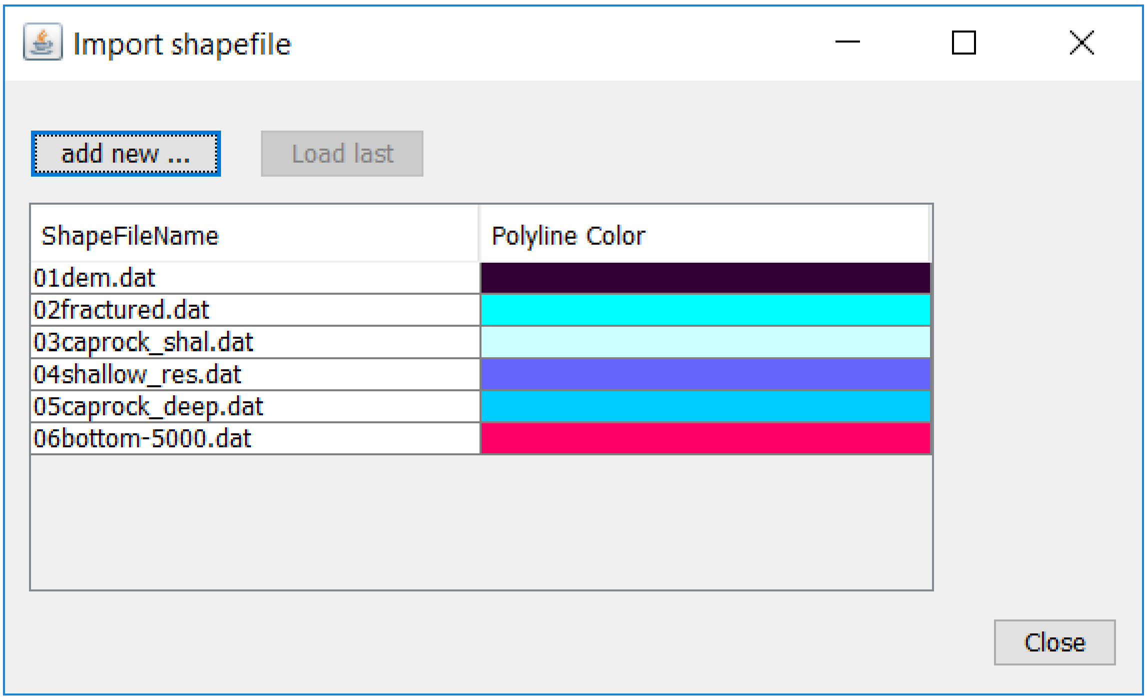


Fig. 74 – ImportShapeFile window after surface file importing

Optionally, user can modify the aspect ratio of the model and of the surface shape by selecting from the menu **Options->3D Block model** object a Z Stretch Factor=5, then click **apply (**Fig. 75**)**.

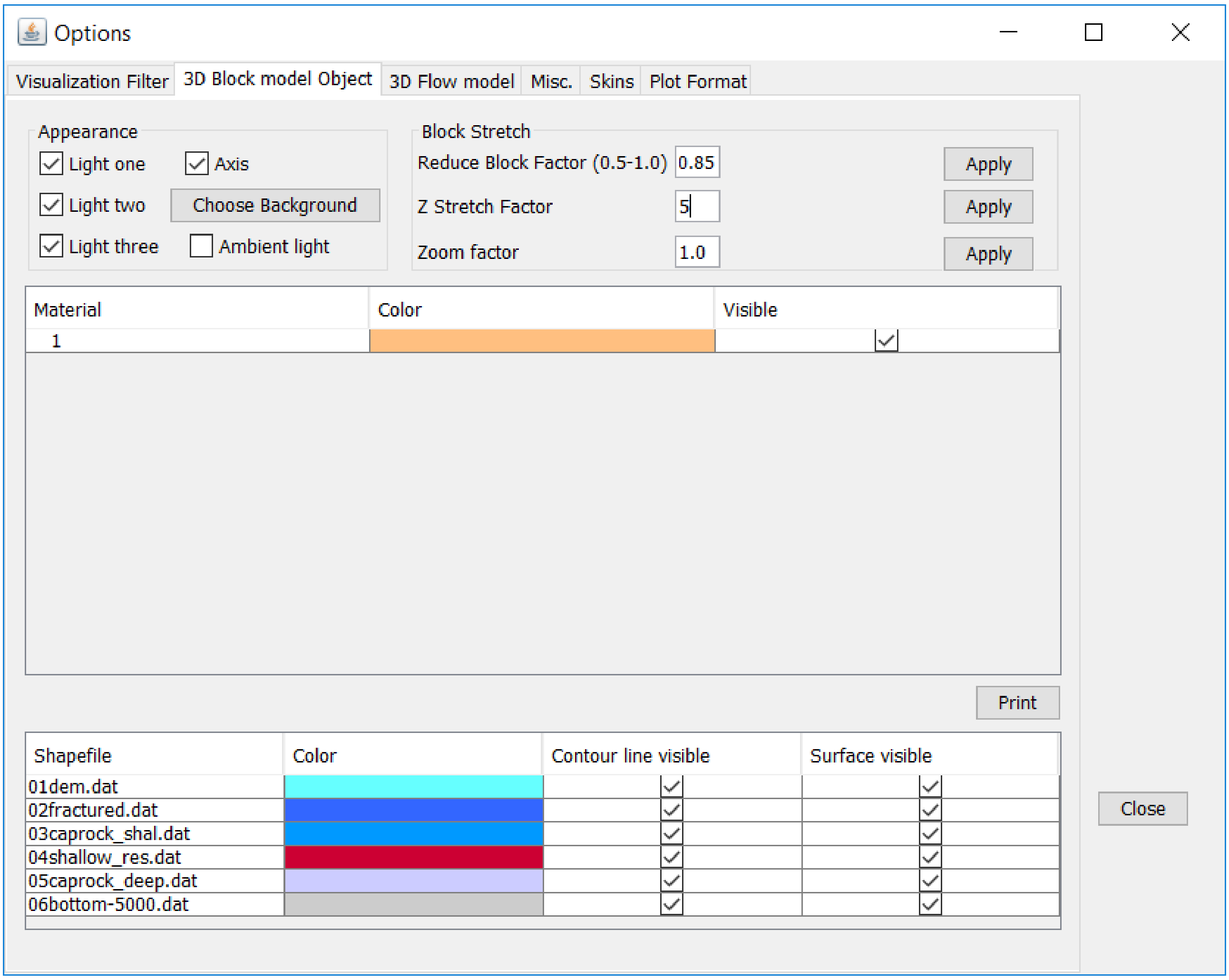


Fig. 75 – Options dialog window

From the View menu, select **View->3D Block model**. The initial view is the top view. Rotate the model and obtain the representation as shown in Fig. 76

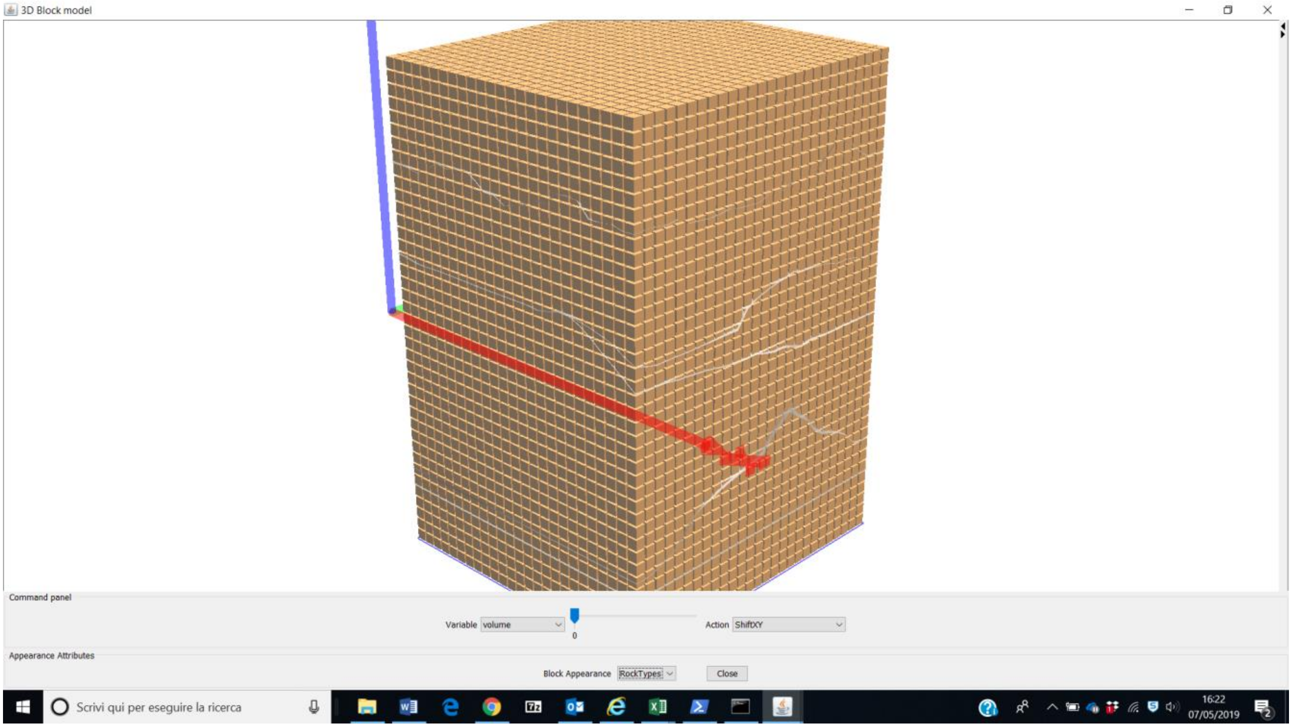


Fig. 76 – The empty 3D structured model and the surfaces

Select from the Action dropdown menu the **AdvancedBlockSelection** command.
From the **AdvancedBlockSelection** dialog window (Fig. 77), select **above** and **01dem.dat** surface selection. Press **Add**.

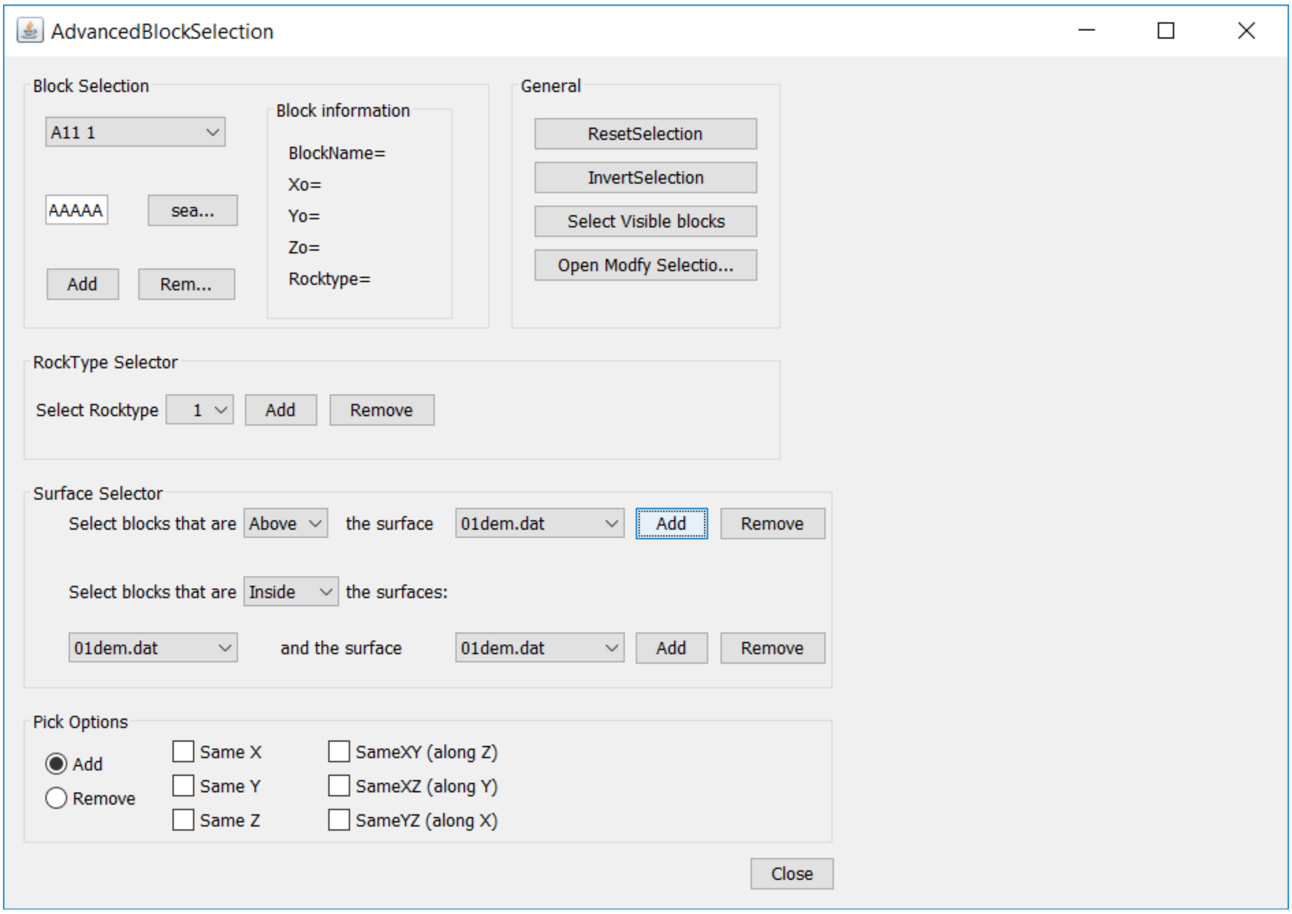


Fig. 77 – Advanced Block Selection dialog window

All the blocks that have the coordinates block node **above** the selected surface will be selected.
Press the **Open Modify Selection**… button.
On the **Properties modifier** dialog window (Fig. 78) insert using the **New Rocktype** text box a new rocktype name “**ATMO1”.** Then click **Apply.**

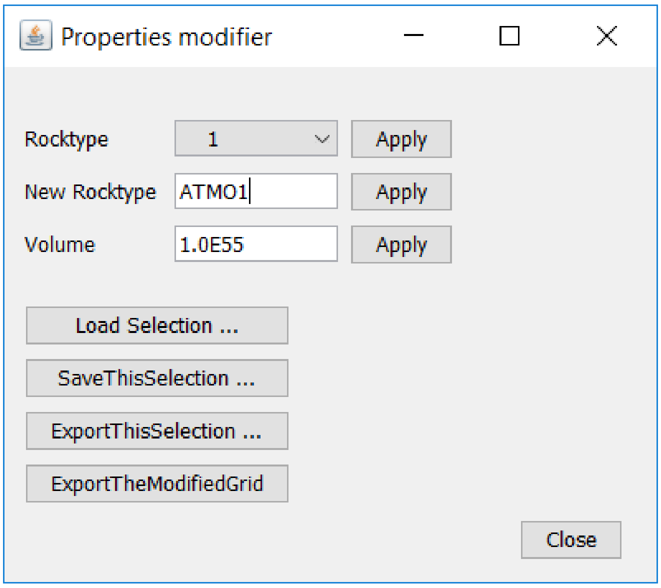


Fig. 78 – properties modifier dialog window

Close the **Properties modifier** dialog window. In the **AdvancedBlockSelection** dialog window press **ResetSelection** and confirm.

The model rocktypes has then been modified as shown in Fig. 79.

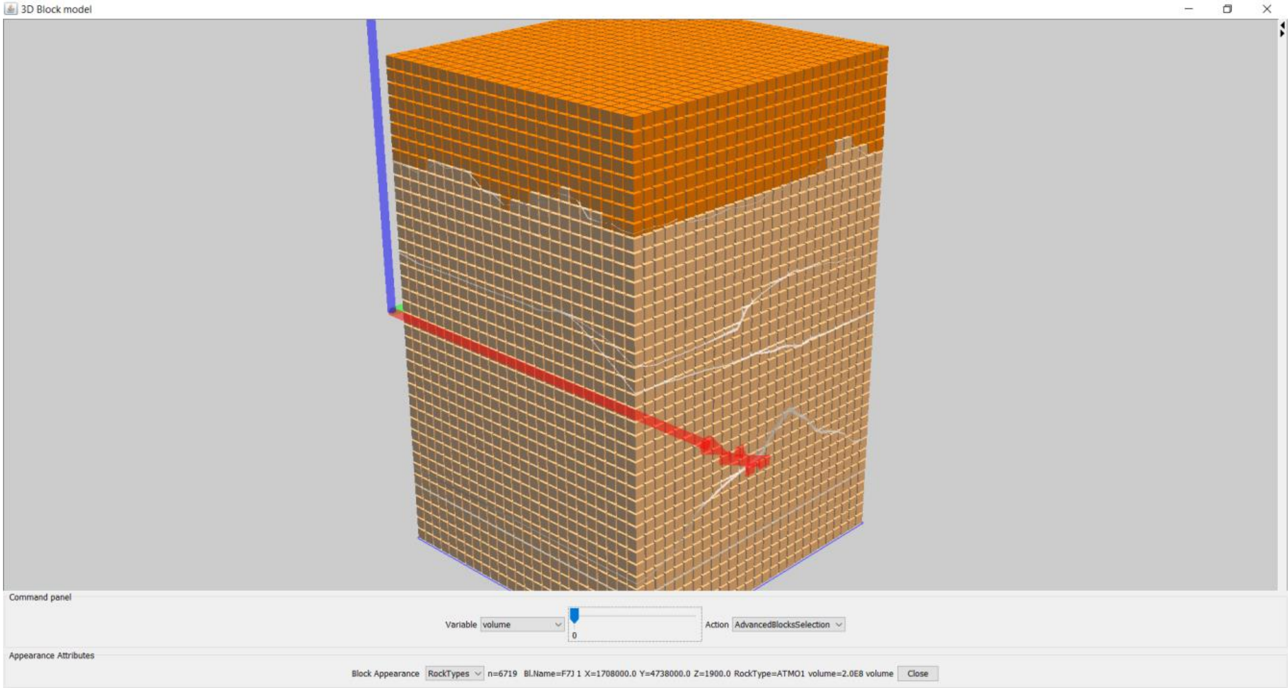


Fig. 79 – The modified model after rocktype assignation

Select now the **Inside** dropdown menu block selector from the **AdvancedBlockSelection** dialog window, panel “surface selector” and select as first surface **01dem.dat** and as the second surface the **02fractured.dat** record. Note that the selected block are hidden from the **ATMO1** type blocks. Open the **Options** dialog window and hide the ATMO1 blocks type and, optionally, the 01dem.dat surface. Open the **Properties modifier** dialog window and assign the material **VULC2**.
By repeating these steps, assign the materials types to each strata, up to have a material type assignation like:

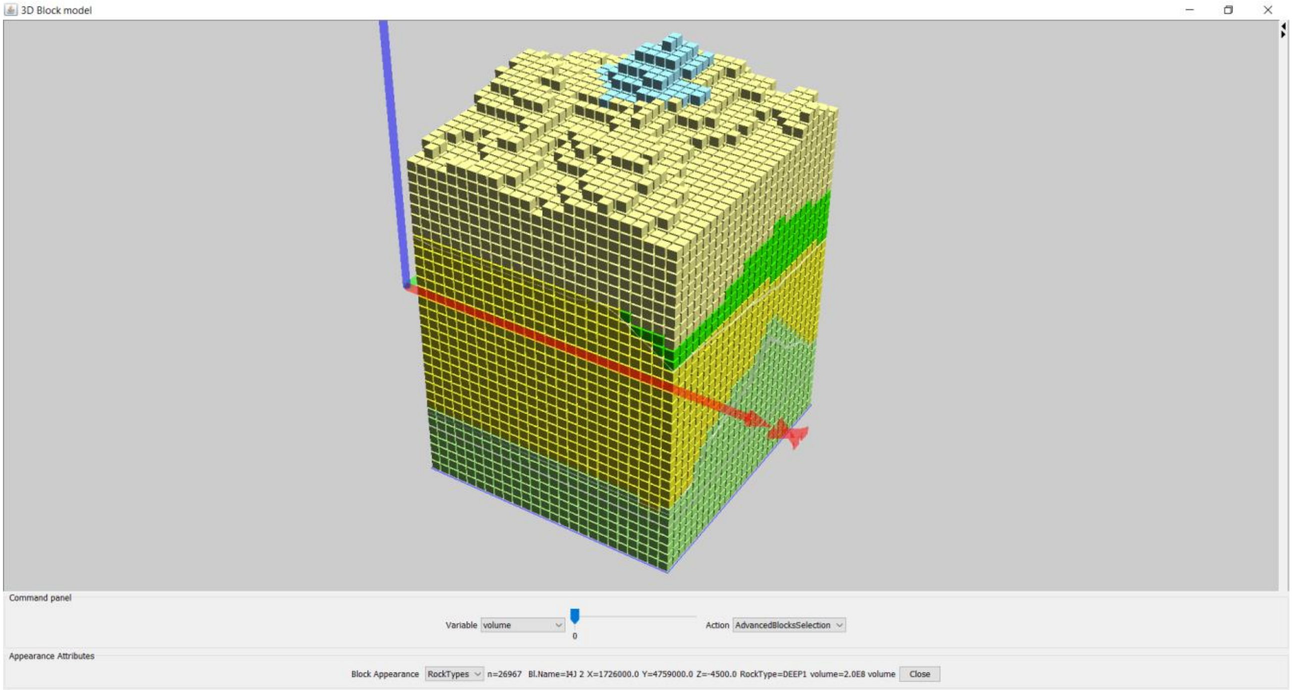


Fig. 80 – The final 3D structured model with the desired rocktype assignation.

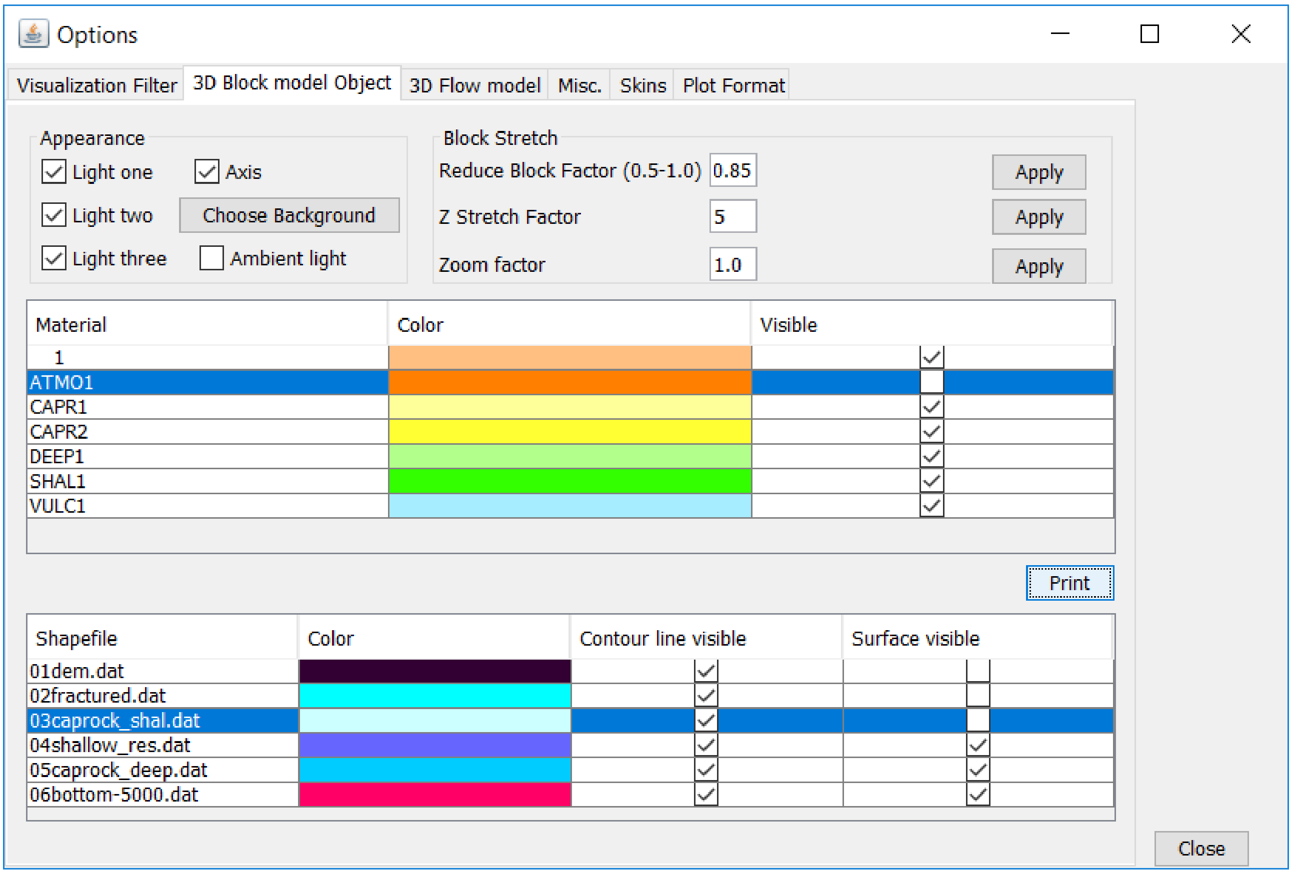


Fig. 81 – The Option dialog window. Note the new rocktypes created during the model modifications.

User may also set the boundary condition for ATMO1 blocks type. This operation can be perfomed by selecting the ATMO1 block type using the **AdvancedBlockSelection** dialog window and then

using the properties modifier by pressing the **Volume apply** button. This action will modify the volume of the selected block to an infinite volume (or to a volume set by the user).
Open the **Properties modifier** dialog window and press **ExportTheModifiedGrid** button. In the **Save File** dialog window choose a name for the new grid, e.g. **model01**, and click the **Save** button.
In the selected folder a new MESH and tough2viewer.dat file, named model01.dat and model01_tough2viewer.dat respectively has been created.
User can open the new generated model again by TOUGH2Viewer in another session for grid inspection before running a TOUGH2 simulation.
Note that:
The strata **SHAL1** have some pinch-out because the distance between the 2 surfaces was near to zero and there are not blocks with the node included in these two surfaces.

6.6 The structured grid case

In this example will be show how to handle material assignation of a pre-existing numerical “empty model” using a set of raster surfaces.
In the folder \\ExampleData\ 06_Structured2D\ you will find the following sub-folders:

- model;
- surfaces;
- voro2mesh;

The voro2mesh folder contains all the input data needed for the generation of the empty model, using VORO2MESH. For details about VORO2MESH, please see Bonduà et al. (2017).
In the model folder, you will find the MESH file and the tough2viewer.dat file (geometrical data) as generated by VORO2MESH.
The surfaces folder contain the surfaces files that will be used for rocktype assignation, in raster and in ply file format:

- 00top.dat;
- 00top.ply;
- 01top.dat;
- 01top.ply;
- 02top.dat;
- 02top.ply;
- 03top.dat;
- 03top.ply;

A graphical representation of these surfaces is given in Fig. 82.

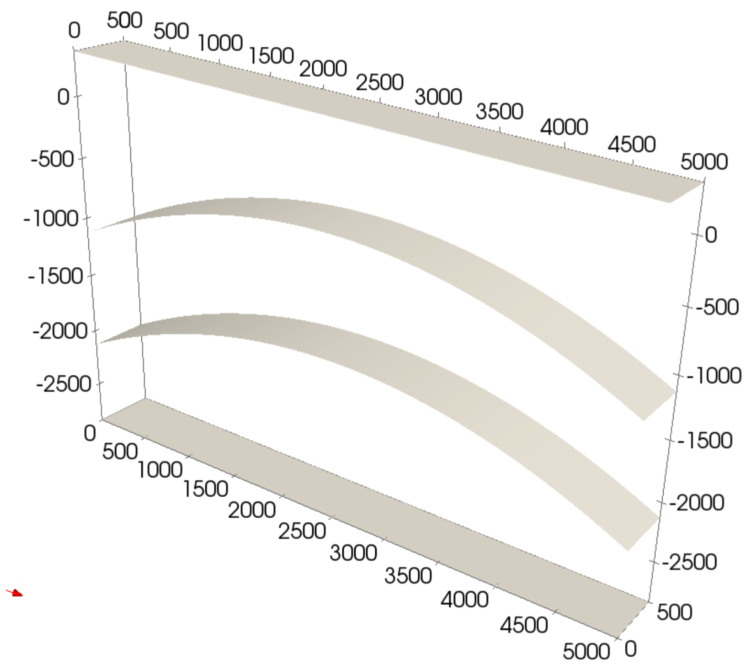


Fig. 82 – The four surfaces used to define the several geological horizons. Image obtained with Paraview.

Each file represent the top surface of a different geological layer. In this presentation we just focus out attention on the rocktype assignation, without considering petrophysical properties of the geological layer, to be treated separately in the ROCKTYPE section of the TOUGH2 input file.
The limits of the domain of the numerical model are (m):

x_max=+5000.0
x_min=0.0
y_max=+1000.0
y_min=0.0
z_max=500.0
z_min=-3000.0

The dimension of the blocks are: 100x1000x100 m^3.
Open TOUGH2Viewer and from the **File** menu select **Open unstructured V++ grid…**
Select the tough2viewer.dat file and the MESH file from the path: \\ExampleData\06_Structured2D\model.
Press the **CreateEmptyModel** button and then close the dialog by selecting the **Close** button.
From the **File** menu select **Import Surface File…**
Select the files in the following order:

- 00top.dat;
- 01top.dat;
- 02top.dat;
- 03top.dat;

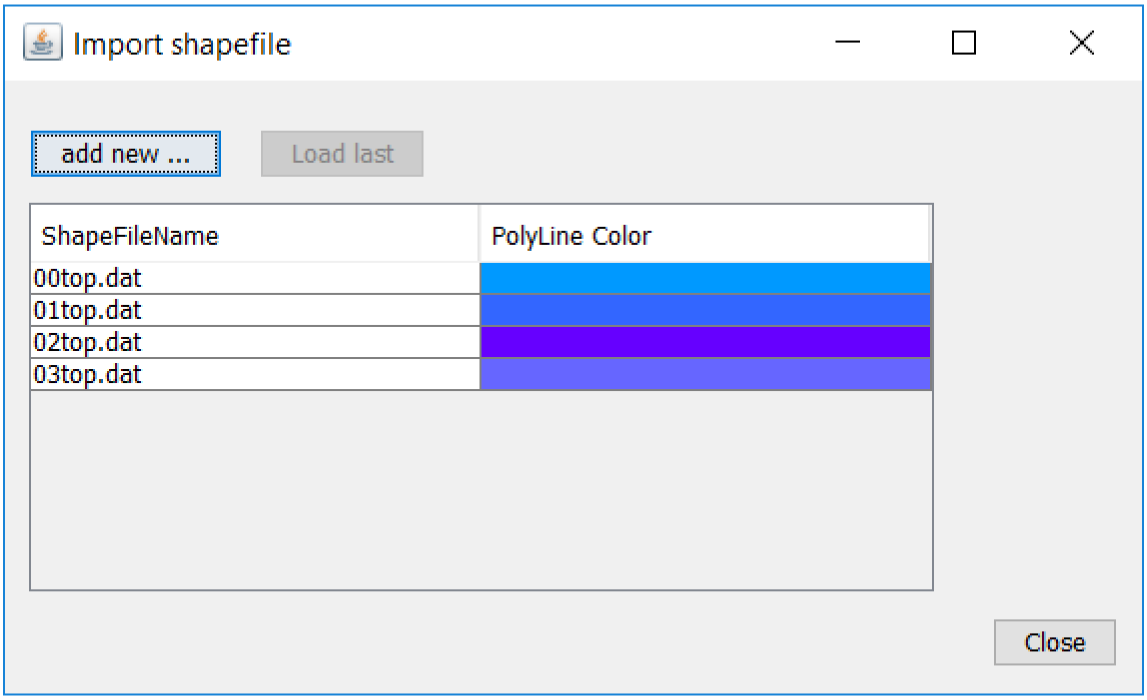


Fig. 83 – The ImportShapefile dialog window after data loading.

From the TOUGH2Viewer View menu, select **View->3D Block model**. From the dropdown **BlockAppearance,** select the **Rocktype** item**.** The initial view is the top view.
Rotate the model and obtain the representation as shown in Fig. 84.

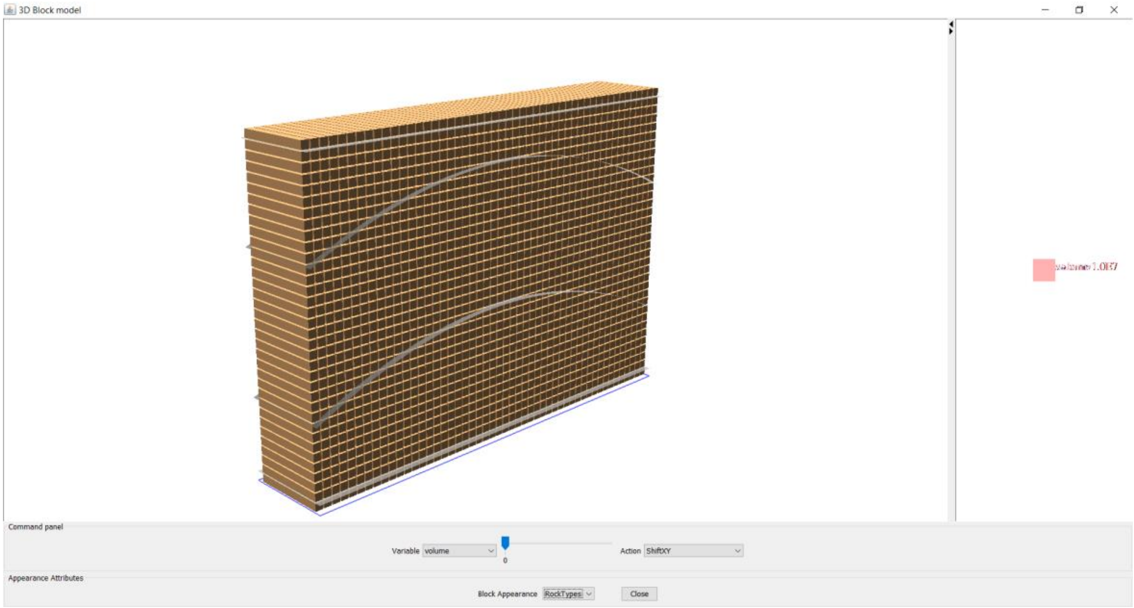


Fig. 84 – The model and the surfaces visualized in the Model3D window.

From the **Action** menu, select the **AdvanceBlockSelection** command. The AdvanceBlockSelection dialog window of Fig. 85 will be shown.

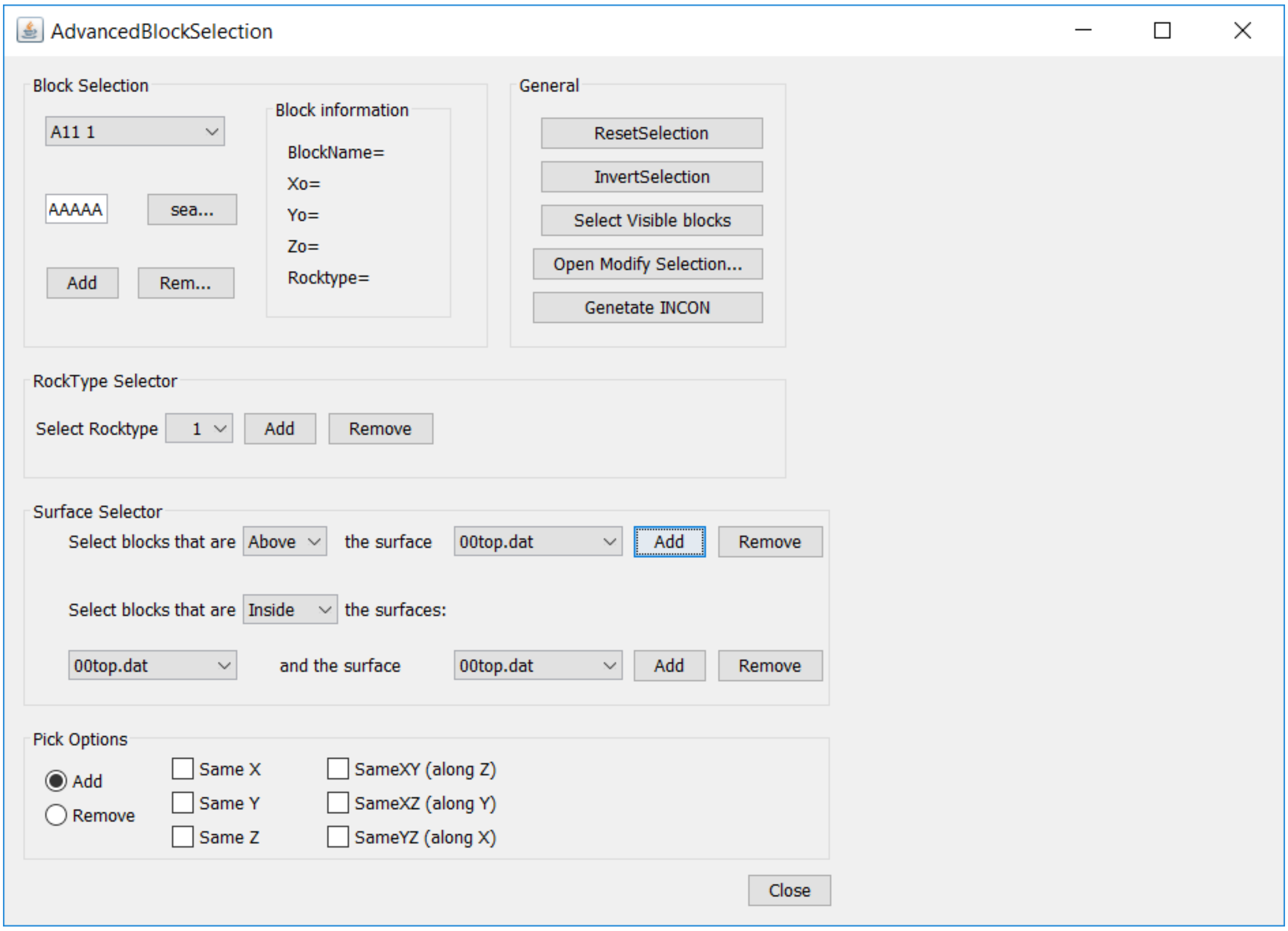


Fig. 85 – The AdvanceBlockSelection dialog window.

From the Surface Selector panel, press the **add** button that belong to “Select blocks that are above the surface 00top.dat” line.

With this command all blocks that are above the surface 00top.dat will be selected. Press now the **Open Modify selection…** button. In the dialog window, enter a new rock type called ATMO1. Then press **Apply** (see Fig. 86).

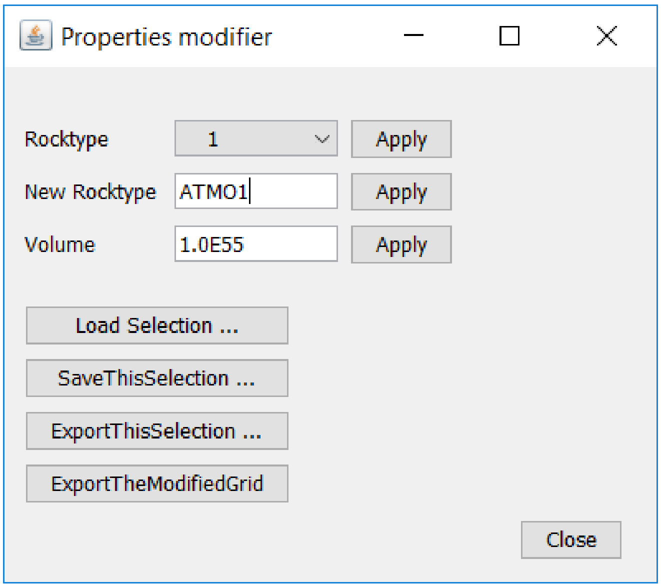


Fig. 86 – properties modifier dialog window.

Close the window and in the **AdvancedBlockSelection** press the **ResetSelection** button.
The model will now appear as shown in Fig. 87.
In the Surface Selector panel of the **AdvancedBlockSelection** dialog window, press the **add** button relatively to the command "Select blocks that are inside the surface 00top.dat and the surface 01top,dat". By the **Open Modify Selection** command, assign to the selected blocks a new rocktype **CAPR2**.
The model is now modified as shown in Fig. 89.
By repeating the above procedure by changing the surfaces, a final model can be obtained.
A view of the final model is shown in Fig. 90.

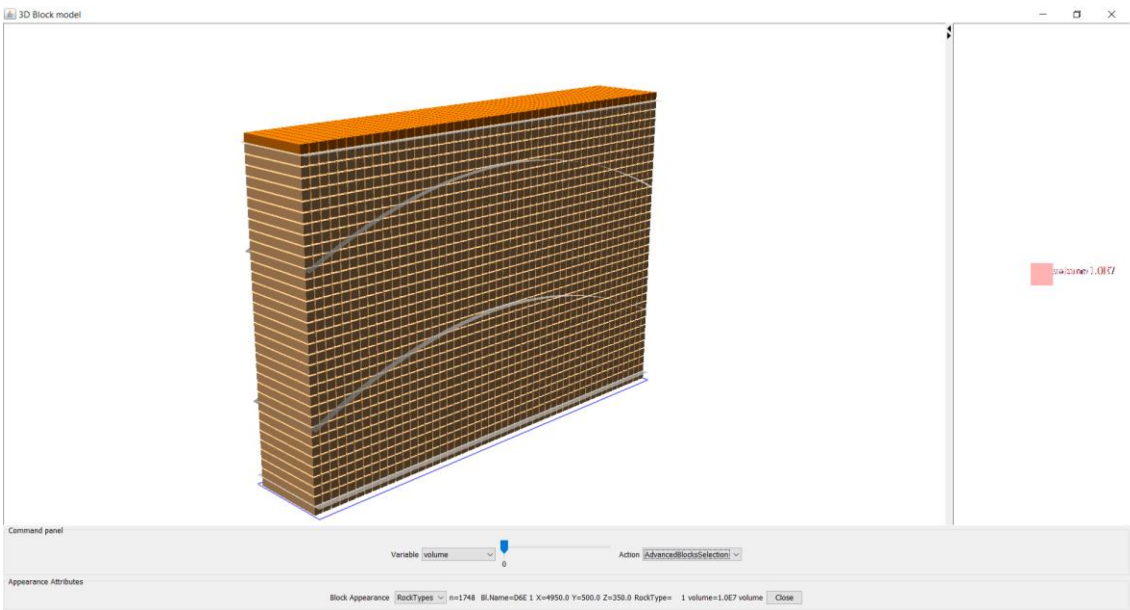


Fig. 87 – The modified grid.

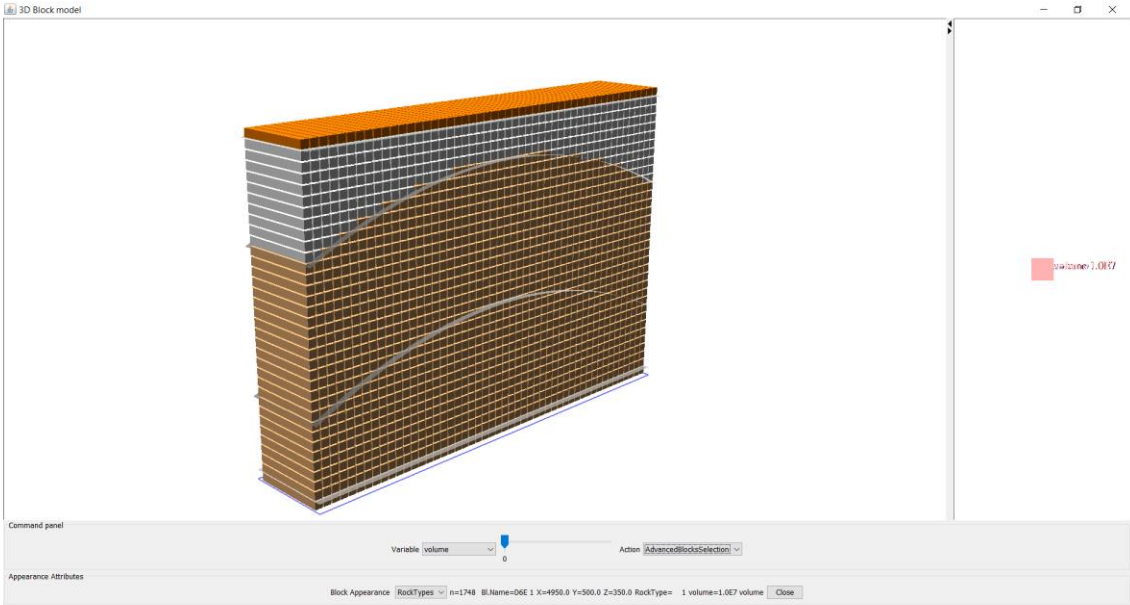


Fig. 88 – The block selection between two surfaces.

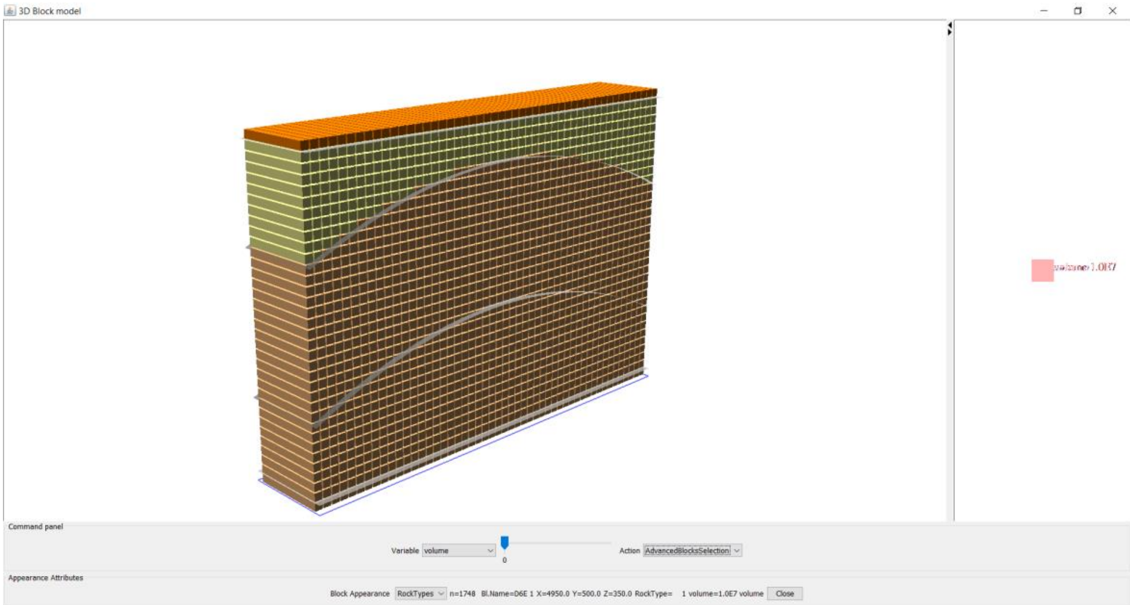


Fig. 89 – The modified grid.

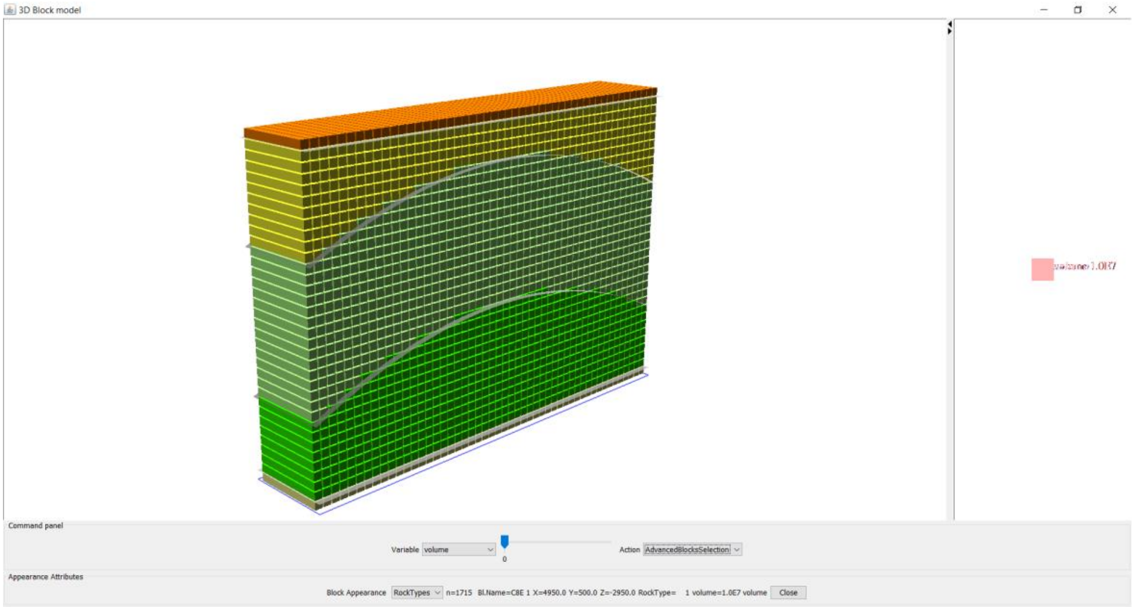


Fig. 90 – The final grid.

User can change the rocktype color by the **Tool->Option** dialog window. In the Fig. 91 is shown the same model of Fig. 90 with the color of rocktypes changed and the surfaces colored by rocktype. Also a setVOI command is applied to the visualization.

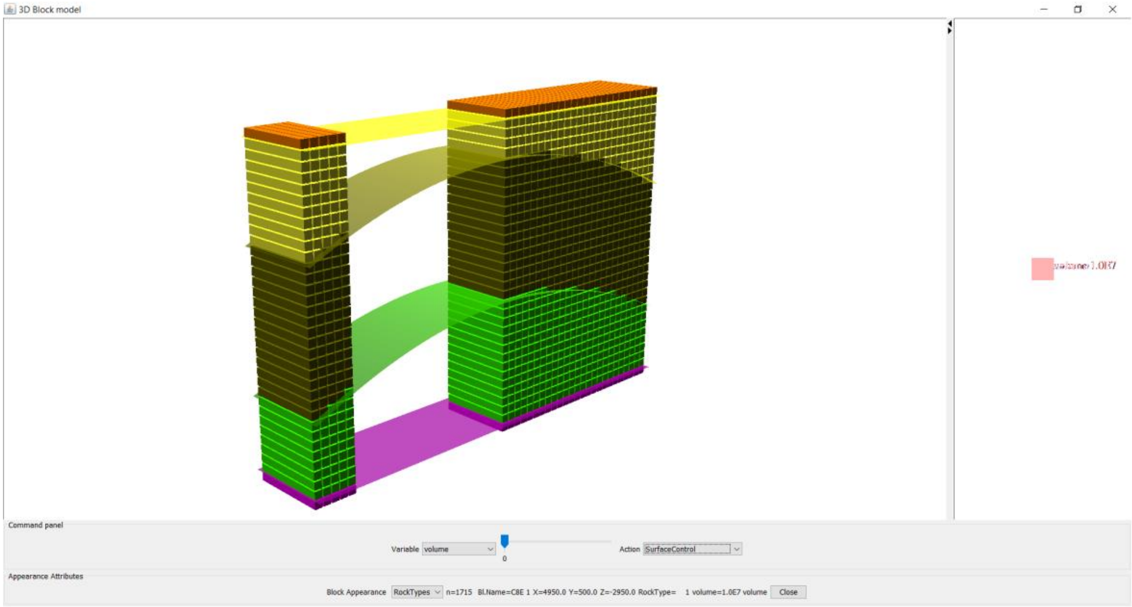


Fig. 91 – The final model with modified colors and VOI applied.

After rocktype assignation and/or blocks volume editing, the modified model must be saved by using the **ExportTheModfiedGrid** command of the **PropertiesModifer** dialog window. In Fig. 92 is shown the save dialog window after pressing the **ExportTheModifiedGrid** button. Note that in the same folder will be saved also a copy of the tough2viewer.dat, because in case of a generation of a sub grid, a different geometry file is needed.

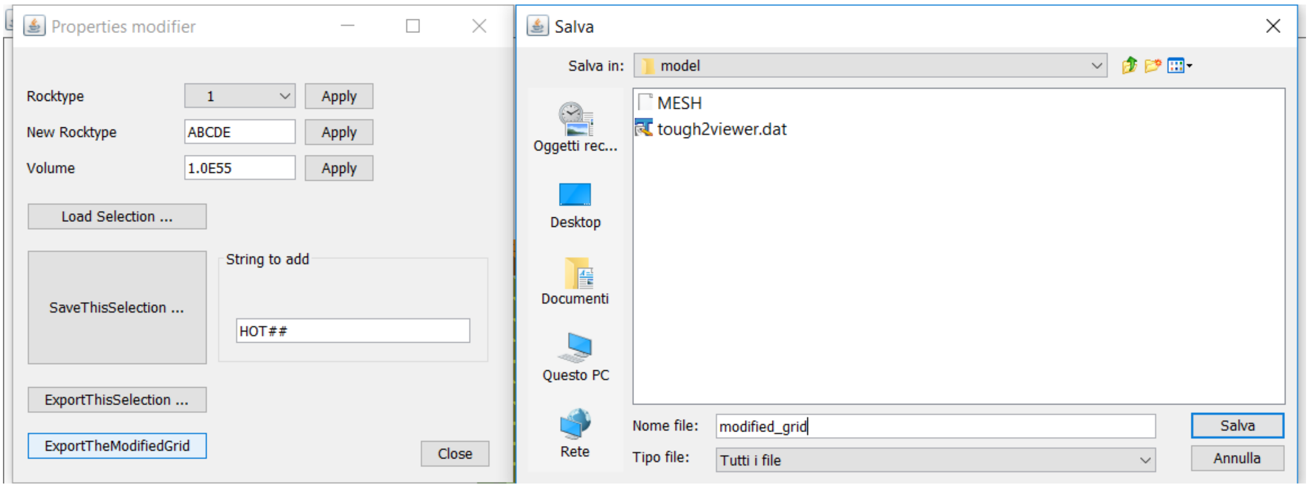


Fig. 92 – The modified grid is saved as “modified_grid”.

INCON generation

Before creating a new INCON file, a previous INCON or SAVE file must be loaded.
Load the INCON.dat file by the **Open Unstructured V++ grid…** dialog window.
In the **AdvancedBlockSelection** dialog window, press the **Generate INCON** button or, alternatively, from the TOUGH2Viewer main menu **select Tools->Create INCON**. The **INCON Generato**r dialog window will be shown. From the variable selector select INCON_0 and then press the **add** button. Repeat the previous step up to the INCON_3 variable. Select from the **INCON VARIABLES** selector the INCON_0 variable and update the plot by pressing the **UpdatePlot** button. The INCON Generator dialog will now have the aspect of Fig. 93.

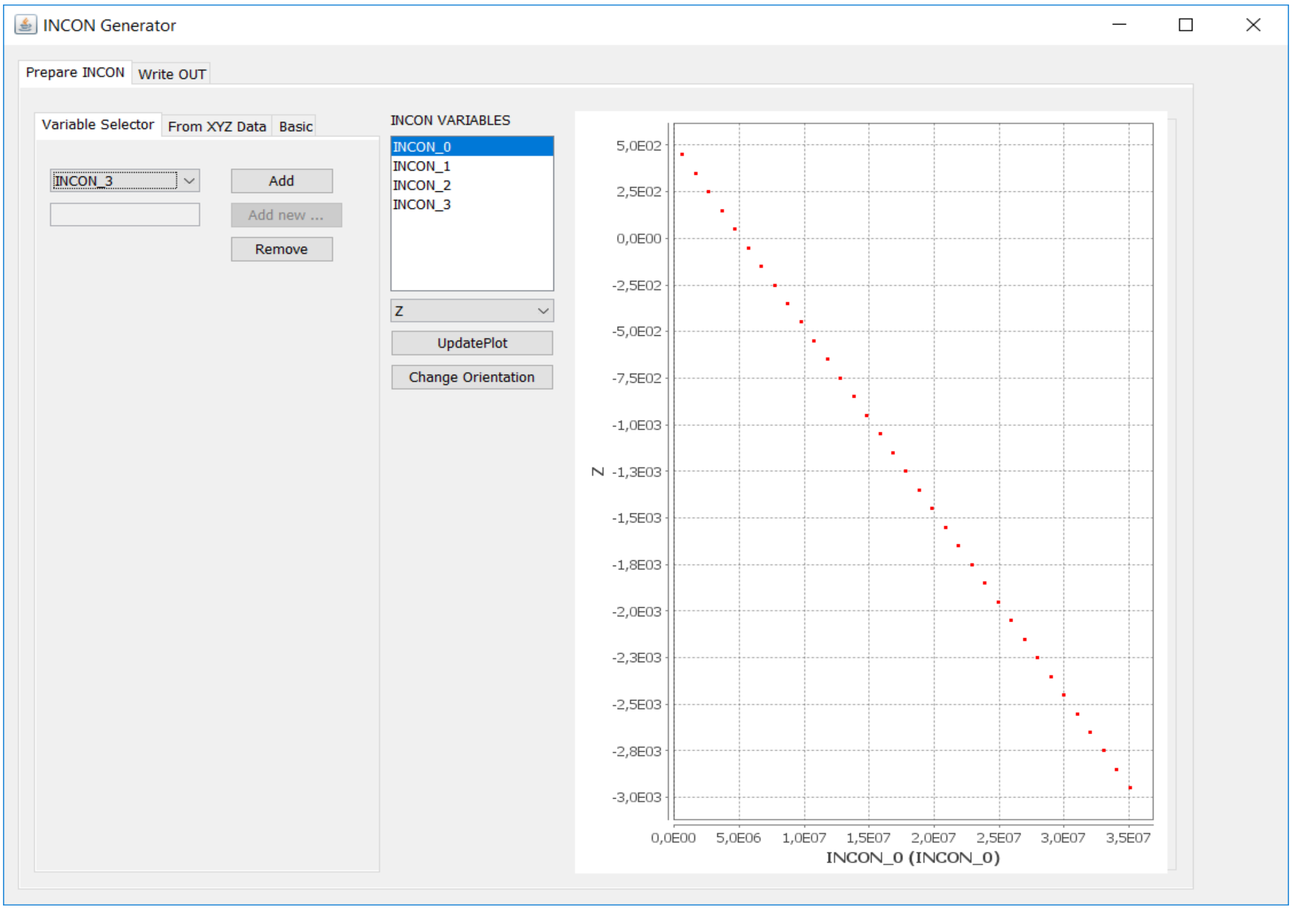


Fig. 93 – The INCON generator dialog window

Select the **basic** tab. This tab allow to build/edit the INCON variable by means of a linear function. Press the P button (Pressure) to load default parameter for hydrostatic condition. Press the **zero to top** button to modify the equation in a manner that the top block have the pressure of the **Zero** text field (by default, atmospheric pressure for the top blocks). Press the **Preview** button to visualize the new INCON before apply changes. The INCON Generator dialog window appear now as shown in Fig. 94.

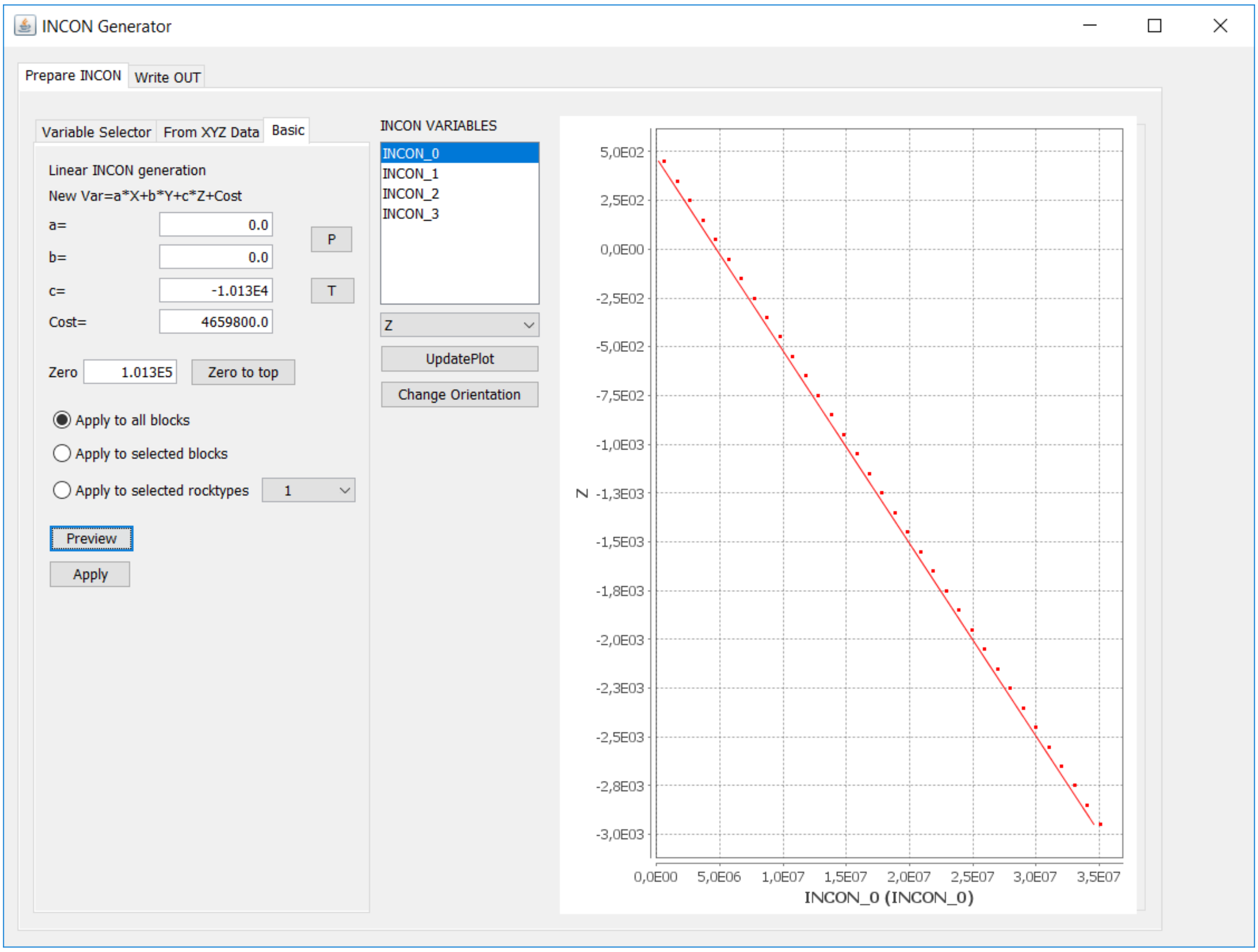


Fig. 94 – The INCON generator dialog window with a preview of the linear function.

After pressing the **Apply** and **Preview** buttons again, the variable are modified as shown Fig. 95. Repeat the previous steps for INCON_1 (temperature), INCON_2 (XCO2) and INCON_3 (XNACL). It worth to mention that the tool allow to apply the changes to all the blocks, to the blocks of a certain rocktype or just to the selected blocks.

Once that the INCON has been modified, user can visualize the new values in to the Model3D visualization window by choosing the variable to be visualized by selecting the variable to plot. To save the edited INCON file, a **write out** tab is designed to obtain the formatted INCON file as required by TOUGH. Press the **write out** tab and the **preview** button. The dialog window now display a preview of the INCON file (see Fig. 96). Note that the numbering format can be changed by the user if the particular EOS used for the numerical simulation require a special format.

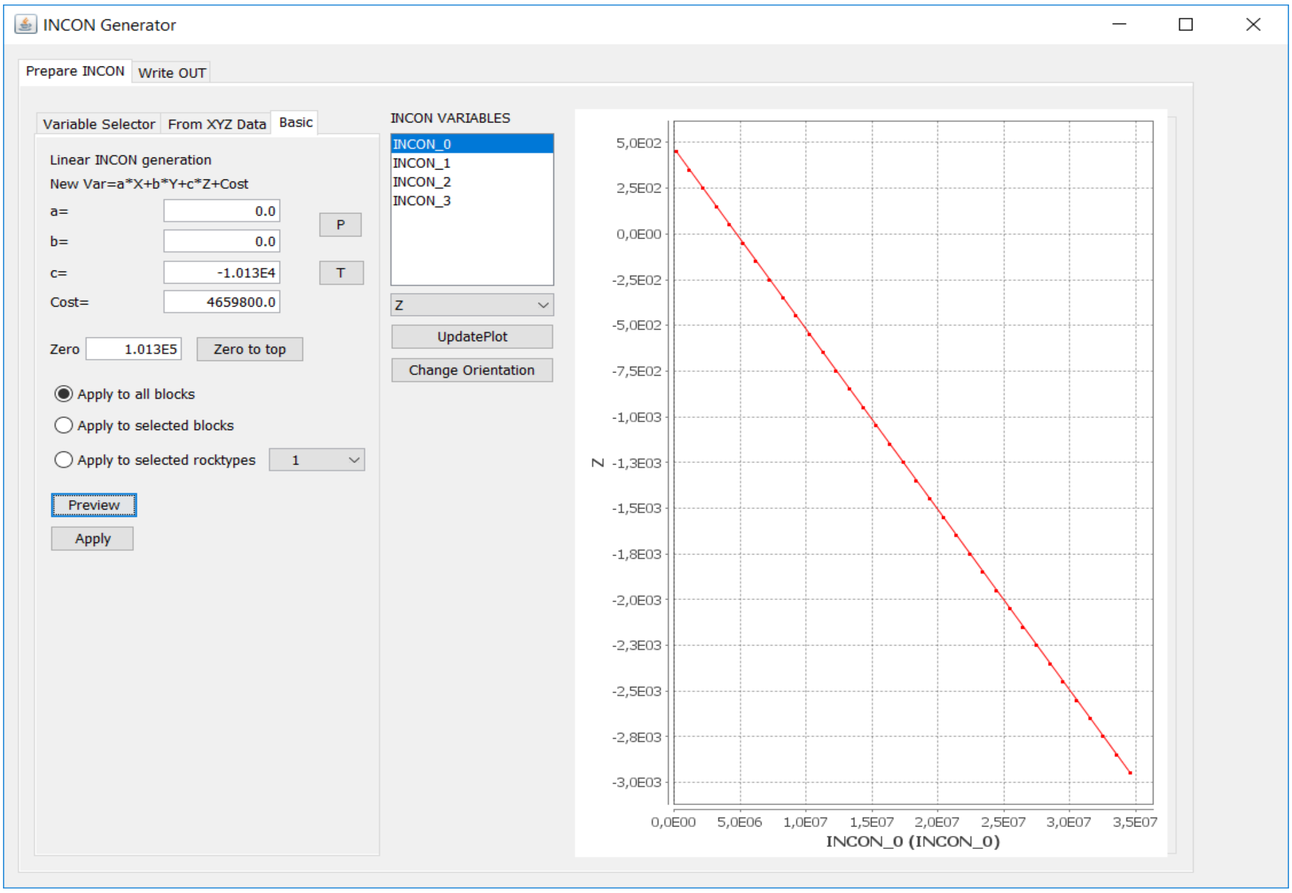


Fig. 95 – The INCON generator dialog window after changes applied.

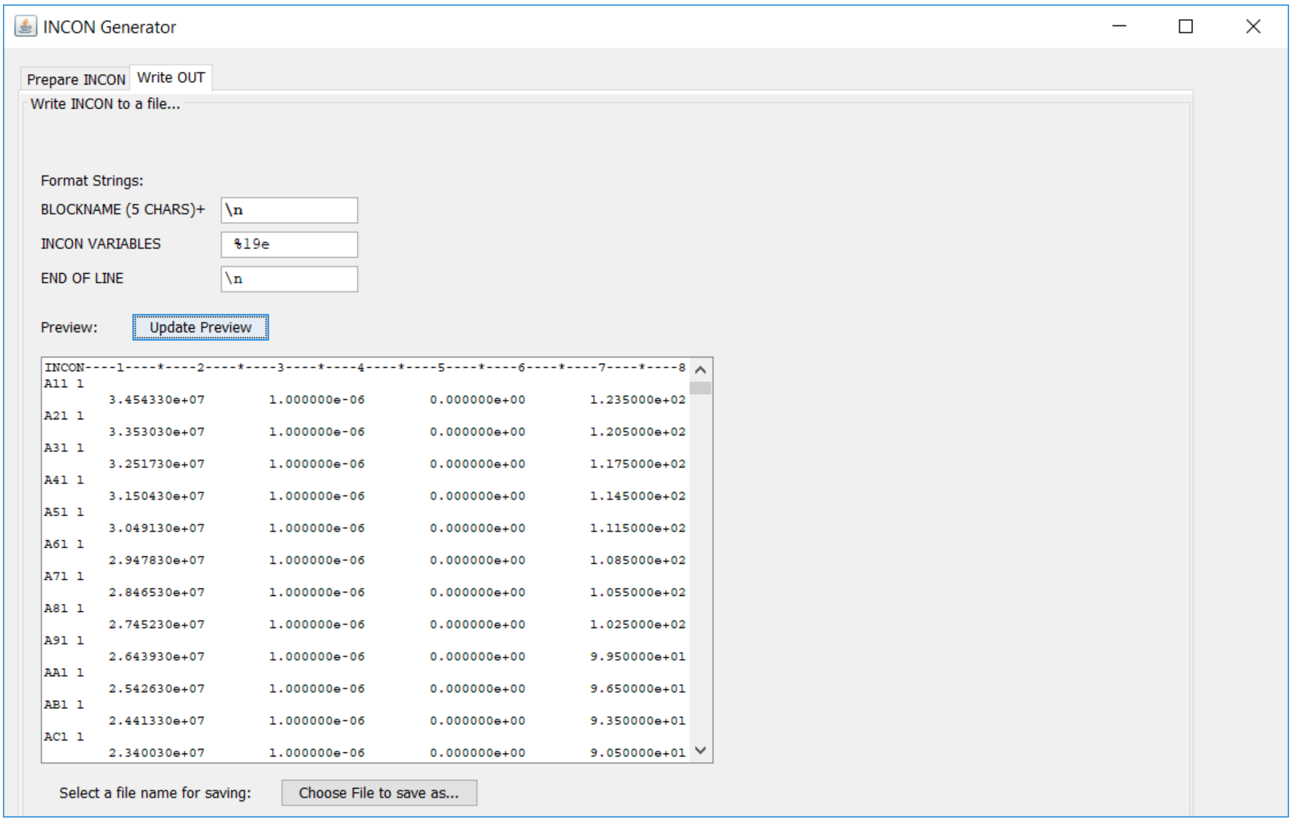


Fig. 96 – The INCON generator preview dialog window.

As last step, press the **Choose File to save as…** button to generate the new INCON file. The created file will be [inserted name file].INCON.
The generated files (the MESH and the INCON file) are now ready to be used in a TOUGH simulation.
It worth to mention that the number of primary variables written in to the INCON file have to match with the number of variables required by the specific EOS - TOUGH simulator and option, and no-check or validation will be performed on it by TOUGH2Viewer.

7 References

Battistelli, A, Calore, C., Pruess K., 1997. The simulator TOUGH2/EWASG for modelling geothermal reservoirs with brines and a non-condensible gas. Geothermics 26(4):437–464.

Berry, P., Bonduá, S., Bortolotti, V., Cormio, C., Vasini, E.M., 2014. A GIS-based open source pre-processor for georesources numerical modeling, Environmental Modelling & Software, 62, 52-64, ISSN 1364-8152.

Bonduá, S., Berry, P., Bortolotti, V., Cormio, C. (2012). TOUGH2Viewer: A post-processing tool for interactive 3D visualization of locally refined unstructured grids for TOUGH2. Computers & Geosciences, 46, p. 107-118.

Bonduà, S.; Battistelli, A.; Berry, P.; Bortolotti, V.; Consonni, A.; Cormio, C.; Geloni, C.; Vasini, E. M., (2017). 3D Voronoi grid dedicated software for modeling gas migration in deep layered sedimentary formations with TOUGH2-TMGAS, COMPUTERS & GEOSCIENCES, *in press*.

Pruess, K., Oldenburg, C., Moridis, G., 1999. TOUGH2 User's Guide, Version 2.0, Report LBNL-43134. Lawrence Berkeley National Laboratory, Berkeley, California, (Report LBNL-43134).

8 List of figures

Fig. 1 – Java Folder....4
Fig. 2 – j3dcore-ogl.dll location....5
Fig. 3 – j3dcore.jar, j3dutil.jar and vecmath.jar folder....5
Fig. 4 – (a) TOUGH2Viewer main dialog (b) File menu items....9
Fig. 5 – TOUGH2Viewer View menu....10
Fig. 6 – TOUGH2Viewer Analysis menu....10
Fig. 7 – TOUGH2Viewer Tools menu....11
Fig. 8 - The TOUGH2Viewer empty main window....12
Fig. 9 - The "Open structured grid" dialog window....13
Fig. 10 - Main window with geometric information of the loaded grid....13
Fig. 11 - 3D Default representation of the Model, Top View....14
Fig. 12 - 3D representation of the model, FrontView....14
Fig. 13 - Pressure contour map....15
Fig. 14 - Temperature contour map....15
Fig. 15 - Vector map of the heat flow....16
Fig. 16 - Vector map of the variable VEL(LIQ.)....16
Fig. 17 - Min and Max value for simulation data. The default plot is first variable vs time....17
Fig. 18 - Open Structured grid dialog....18
Fig. 19 - Open unstructured V++ grid dialog window....19
Fig. 20 - 3D Block model window, top view....20
Fig. 21 - 3D Block model box showing the Temperature variable at the time step 2....20
Fig. 22 - 3D modelFrame horizontally sectioned....21
Fig. 23 – Graph of the pressure versus the x coordinate....21
Fig. 24 – Graph of the pressure versus the z coordinate....22
Fig. 25 – Graph of the pressure versus the z coordinate, with background coloring....22
Fig. 26 - ExpandZ command result....23
Fig. 27 - QuickInfo command result, information are printed in a panel at the bottom of the window....24
Fig. 28 - SetVOI function applied to the model....24
Fig. 29 - 3D Block model in material color scale....25
Fig. 30 - Vector representation of the heat flow....25
Fig. 31 - Vertical heat flow profile....26
Fig. 32 - 3D IsoSurface dialog....26
Fig. 33 - IsoSurface Dialog for pressure and time step 0....27
Fig. 34 - Dialog window for the pressure IsoSurface....28
Fig. 35 - 2D Contour Plot for the Pressure (Pa)....29
Fig. 36 - TOUGH2Viewer empty main window....30
Fig. 37 - Open unstructured grid dialog window....30
Fig. 38 - TOUGH2Viewer main window that summarizes some geometric data of the loaded numerical model....31
Fig. 39 - Import shapefile dialog window....31
Fig. 40 - Dialog window to load shape files....31
Fig. 41 - Dialog window to select the file to be loaded....32
Fig. 42 - 2D Preview of the base_ss.shp file....32
Fig. 43a, b - LoadShpFile Dialog window....33
Fig. 44 - Import shapefile window. The table resume imported shape files....33
Fig. 45 – "Choose a color …" dialog window window....34
Fig. 46 - Top view of the 3D Block model....34
Fig. 47 - SetVOI dialog window....35

Fig. 48 - (a) 3D model partially showed and block selected (white block); (b) information related to the selected block. 35
Fig. 49 - Find Block dialog window. 36
Fig. 50 - Snapshot Preview window dialog window. 36
Fig. 51 - 3D Block Model front view. Material color scale. 37
Fig. 52 - Options dialog window. 37
Fig. 53 - Options dialog with modified parameters. 38
Fig. 54 - 3D Block model. 38
Fig. 55 - Profiles dialog which shows the pressure values along z axis. 39
Fig. 56 - Bottom filtered view of the 3D model with modified color scale. 39
Fig. 57 - 3D Flow vector default viewing. 40
Fig. 58 - 3D Flow model at the time step 1 for the variable VEL(GAS). 40
Fig. 59 - 3D IsoSurface dialog window. 41
Fig. 60 - Isosurface for T=270°C, time step “6”. 41
Fig. 61 – Temperature isolines contour map. 42
Fig. 62 - TOUGH2Viewer empty main window 43
Fig. 63 – Input unstructured grid window 43
Fig. 64 – The main window 44
Fig. 65 - Top view of the 3D Block model. 44
Fig. 66 – View of the SG thermodynamic variables (3D Block model). 45
Fig. 67 – MultipleSelection command 45
Fig. 68 – Properties modifier GUI 46
Fig. 69 – The generated ASCII file 46
Fig. 70 – The modified GENER file 46
Fig. 71 – INCON generator dialog window 47
Fig. 72 – INCON generator dialog window preview 48
Fig. 73 – INCON generator dialog window –write out tab preview 49
Fig. 74 – ImportShapeFile window after surface file importing 51
Fig. 75 – Options dialog window 52
Fig. 76 – The empty 3D structured model and the surfaces 53
Fig. 77 – Advanced Block Selection dialog window 54
Fig. 78 – properties modifier dialog window 54
Fig. 79 – The modified model after rocktype assignation 55
Fig. 80 – The final 3D structured model with the desired rocktype assignation. 56
Fig. 81 – The Option dialog window. Note the new rocktypes created during the model modifications. 56
Fig. 82 – The four surfaces used to define the several geological horizons. Image obtained with Paraview. 58
Fig. 83 – The ImportShapefile dialog window after data loading. 59
Fig. 84 – The model and the surfaces visualized in the Model3D window. 60
Fig. 85 – The AdvanceBlockSelection dialog window. 60
Fig. 86 – properties modifier dialog window. 61
Fig. 87 – The modified model. 61
Fig. 88 – The block selection between two surfaces. 62
Fig. 89 – The modified model. 62
Fig. 90 – The final model. 63
Fig. 91 – The final model with modified colors and VOI applied. 63
Fig. 92 – The modified grid is saved as “modified_grid”. 64
Fig. 93 – The INCON generator dialog window 65
Fig. 94 – The INCON generator dialog window with a preview of the linear function. 66
Fig. 95 – The INCON generator dialog window after changes applied. 67

Fig. 96 – The INCON generator preview dialog window..67