

TOUGH2Viewer

Tutorial

(Version 0.3)

Paolo Berry, Stefano Bonduà, Villiam Bortolotti, Carlo Cormio, Ester Maria Vasini

DICAM

Università di Bologna

www.dicam.unibo.it

Summary

1	Introduction.....	3
2	System Requirements	3
3	Installation, execution and uninstallation instruction.....	4
3.1	TOUGH2Viewer installation.....	4
3.2	TOUGH2Viewer execution.....	5
3.2.1	Linux	5
3.2.2	Windows	5
3.3	TOUGH2Viewer uninstallation.....	5
4	Tutorial Examples	6
4.1	Structured grid model 2D.....	6
4.1.1	Load Files.....	6
4.1.2	3D Block Model view	7
4.1.3	2D Contour Plot visualization	8
4.1.4	2D Flow Vector visualization.....	9
4.1.5	Statistics	10
4.2	Structured 3D Grid model	11
4.2.1	Load Files.....	11
4.2.2	3D block model view	11
4.2.3	3D Flow Vector visualization.....	15
4.2.4	3D Iso Surface	16
4.2.5	2D Contour Plot visualization	18
4.3	Unstructured 3D grid model.....	19
4.3.1	Load Files.....	19
4.3.2	3D Block model view	24
4.3.3	3D Flow Vector visualization.....	29
4.3.4	3D Iso Surface	30
4.3.5	2D Contour map	31
4.3.6	Statistics	32
5	List of figures	33

1 Introduction

TOUGH2Viewer is a Java program capable of displaying unstructured (Voronoi complying) grids, locally refined and structured grids (complying with the MESHMAKER iTOUGH2 format).

In particular, TOUGH2Viewer allows to navigate through a 3D grid compatible with the iTOUGH2 data file format and see:

- 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, of a selected set of blocks, long a Cartesian directions;
- 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, three driven examples will be presented:

1. structured 2D grid generated with MESHMAKER.
2. structured 3D grid generated with MESHMAKER.
3. unstructured 3D grid generated with AMESH.

All the example cases was obtained with the module EWASG as implemented in iTOUGH2.

2 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.

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 Installation, execution and uninstallation instruction

3.1 TOUGH2Viewer installation

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 examples and, once finished copying, unzip it. The unpacked package of files consists of two subdirectories, **dist** (standing for distribution) and **ExampleData**. 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
|   | iTough2Viewer.jar
|   | launch.html
|   | launch.jnlp
|   | License.txt
|   | README.TXT
|   | splash.jpg
|   |
|   \---lib
|       AbsoluteLayout.jar
|       appframework-1.0.3.jar
|       geoapi-2.3-M1.jar
|       geoapi-pending-2.3-M1.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
|       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
| swing-layout-1.0.4.jar
| swing-worker-1.1.jar
| swtgraphics2d.jar
| visad.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
```

3.2 TOUGH2Viewer execution

3.2.1 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 iTough2Viewer.jar [↵]
```

3.2.2 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 iTough2Viewer.jar [↵]
```

3.3 TOUGH2Viewer uninstallation

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

4 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.

Following examples refer to Windows XP operating system.

4.1 Structured grid model 2D

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

Start TOUGH2Viewer, the main window of TOUGH2Viewer will be displayed, as shown in Fig. 1

-.

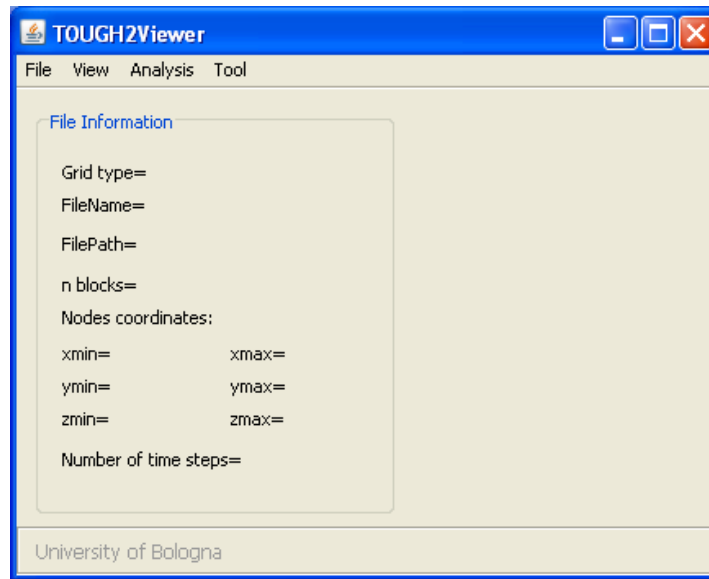


Fig. 1 - The TOUGH2Viewer empty main window

4.1.1 Load Files

From the main menu, select the **File->Open structured grid** menu item to open the "Open structured grid" dialog box (Fig. 2).

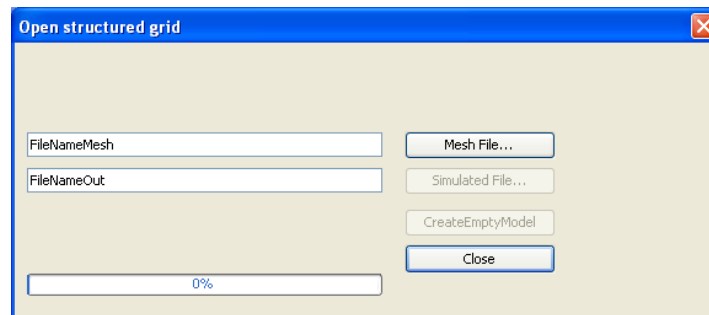


Fig. 2 - The “Open structured grid” dialog box.

Click the **Mesh File** button to display the "Open mesh" dialog box. 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 box. Locate the iTOUGH2 simulation output file "2d.out" and select it 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. 3).

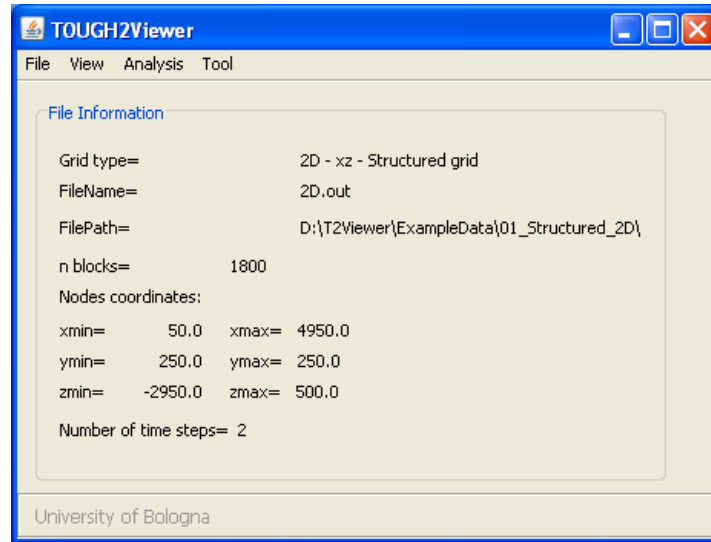


Fig. 3 - Main window with geometric information of the loaded grid.

4.1.2 3D Block Model view

In main menu, click the **View->3D Block Model** menu item, to display the "3D Block Model" dialog box 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. 4. The default variable showed in the dialog box is the first variable reported in the simulation output file (P for pressure in Fig. 4). Default action is ShiftXY.

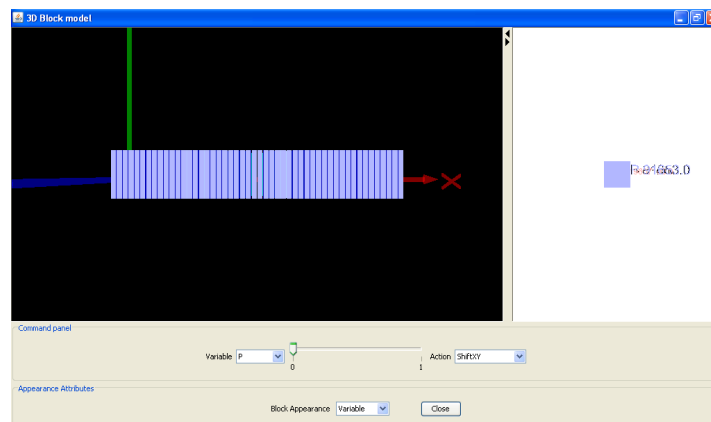


Fig. 4 - 3D Default representation of the Model, Top View.

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

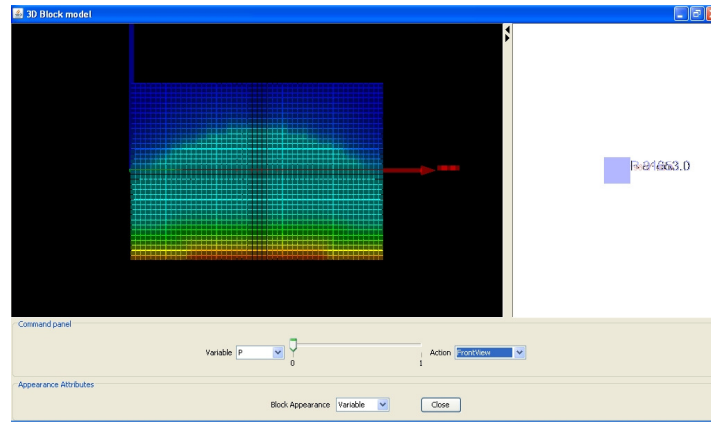



Fig. 5 - 3D representation of the model, FrontView.

Close the 3D Block model dialog by clicking the window icon  or the **Close** button. For more detail on commands available for 3D visualization management, see the example in chapter 4.2 and 4.3.

4.1.3 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 box displays. From the Variable combo box, select **P** (Pressure), then click the **Plot** button (Fig. 6).

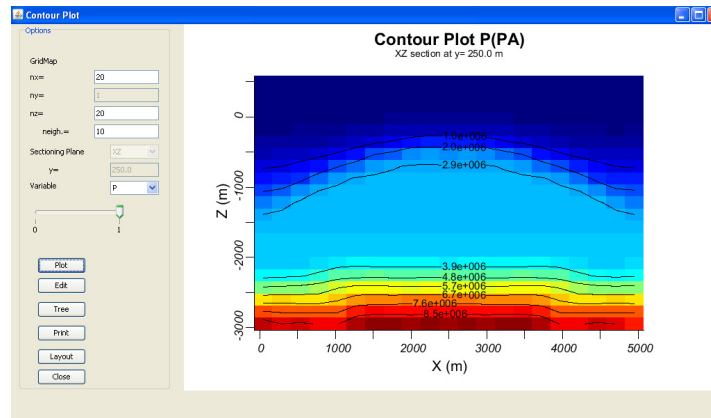


Fig. 6 - Pressure contour map

If one modifies plot's parameter, click the **Plot** button to update the plot in the window (Fig. 7) is shown the contour of the temperature.

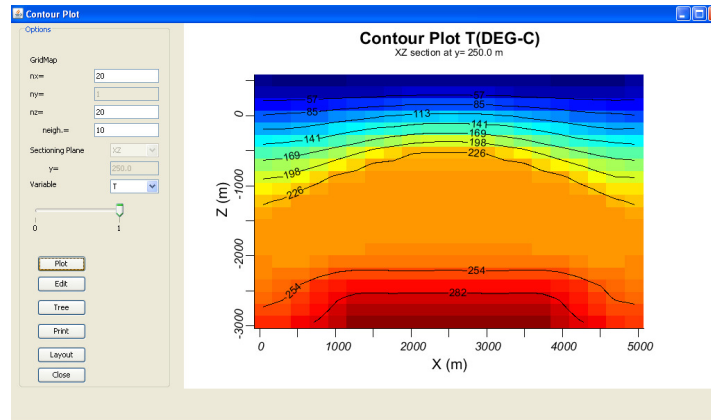



Fig. 7 - Temperature contour map

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

4.1.4 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 box (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).

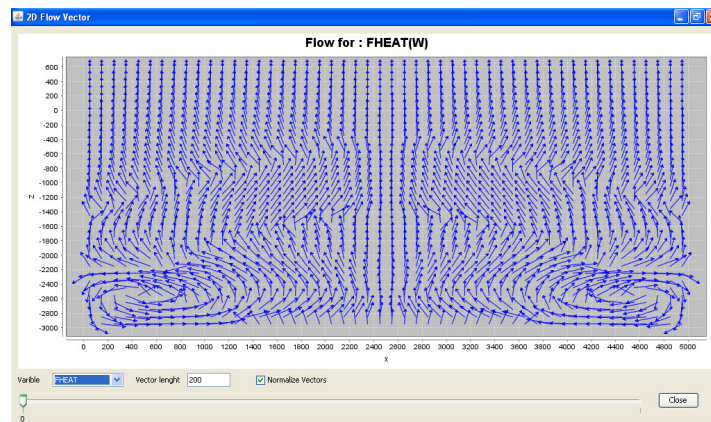


Fig. 8 - Vector map of the heat flow.

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 box now is as shown in Fig. 9.

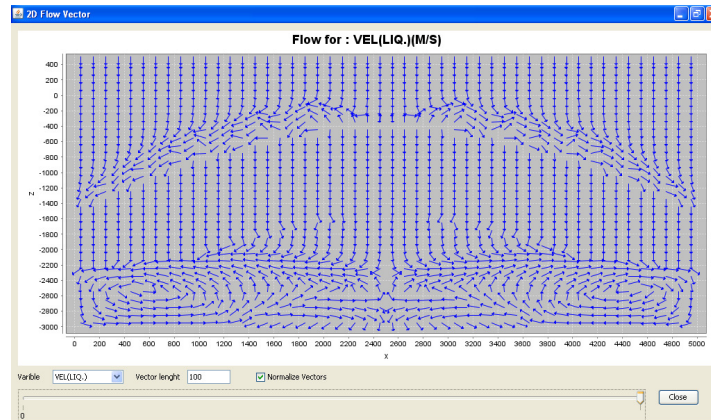



Fig. 9 - Vector map of the variable VEL(LIQ.).

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

4.1.5 Statistics

In the “Statistics” dialog is shown a table with maximum and minimum value for all thermodynamic variables, and it is possible to plot a selectable variable vs simulated time steps. From the menu bar, click the **Analysis->Statistics** menu item. The Statistics dialog box is displayed (Fig. 10).

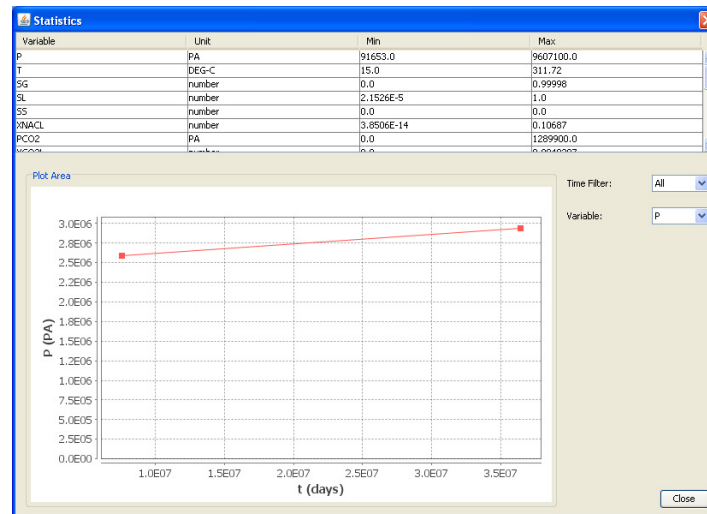



Fig. 10 - Min and Max value for simulation data. The default plot is first variable vs time.

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**.

4.2 Structured 3D Grid model

This example manages a 3D model representing a spatial domain of 5000x5000x3600 m³ discretized in 10x10x36 blocks and simulated for two time steps.

Start TOUGH2Viewer (under Windows OS) by double clicking the **go.bat** batch file.

4.2.1 Load Files

From the TOUGH2Viewer main menu, click the **File->Open structured grid** menu item, and then the dialog box "Open structured grid" is displayed (see Fig. 11).

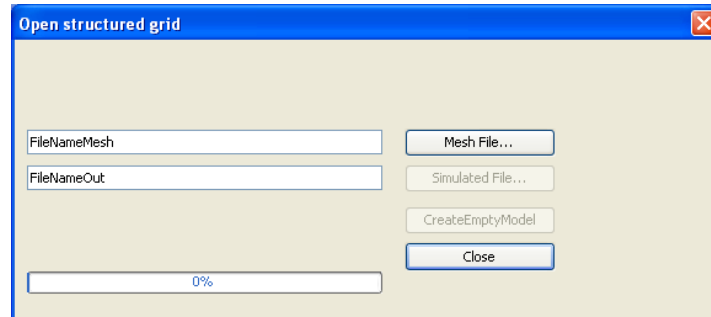


Fig. 11 - Open Structured grid dialog.

Click the **Mesh File** button to display the Open mesh dialog box and load the "MESH.dat" from the "\T2Viewer\ExampleData\02_Structured_3D\" directory.

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 box 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.

4.2.2 3D block model view

From the main menu, click **View -> 3D block model** menu item. The "3D Block model" dialog box is displayed with the default setting, as shown in Fig. 12.

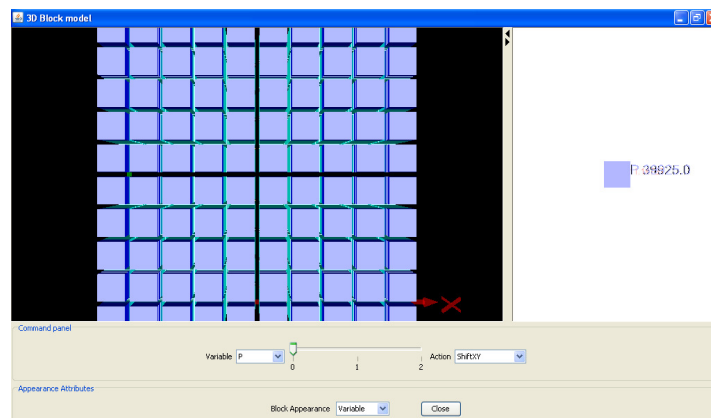


Fig. 12 - 3D Block model window, top view.

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 box, will be similar to that show in Fig. 13.

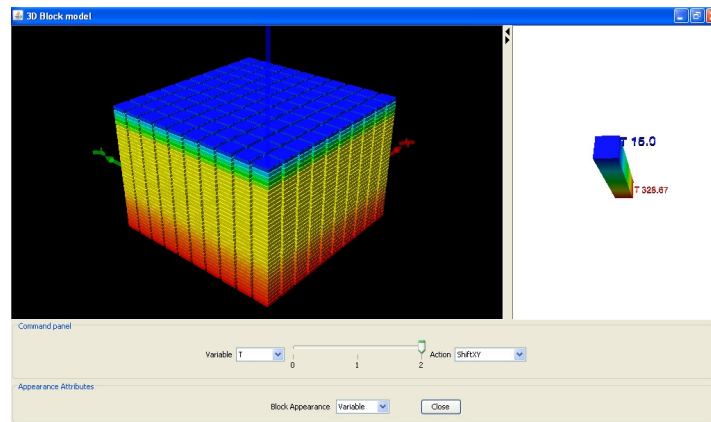


Fig. 13 - 3D Block model box showing the Temperature variable at the time step 2.

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 model will be sectioned as shown in Fig. 14.

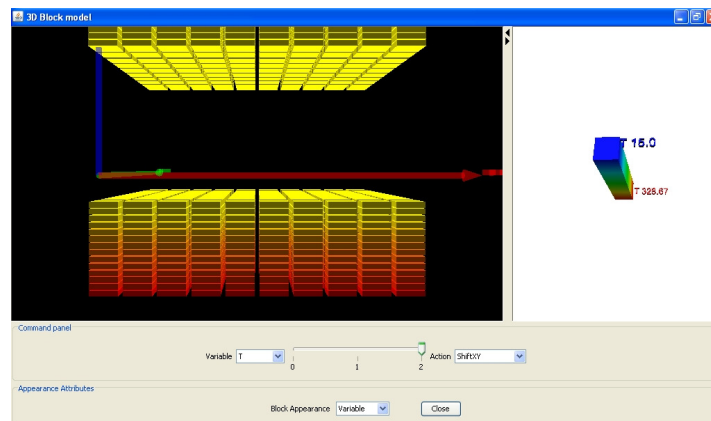


Fig. 14 - 3D modelFrame horizontally sectioned.

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 box is displayed, as showed in Fig. 15 and Fig. 16. 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.

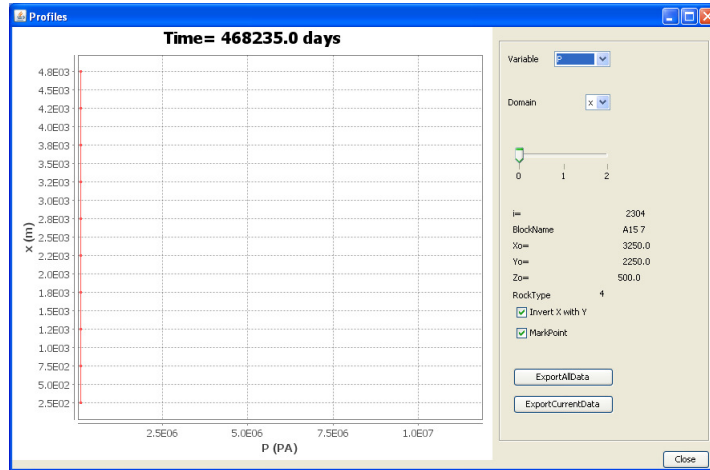


Fig. 15 – Graph of the pressure versus the x coordinate.

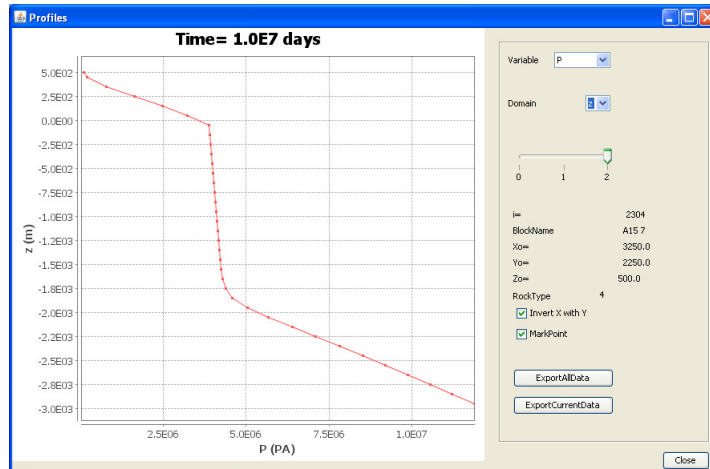



Fig. 16 – Graph of the pressure versus the z coordinate.

Close the **Profiles** dialog box 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. 17).

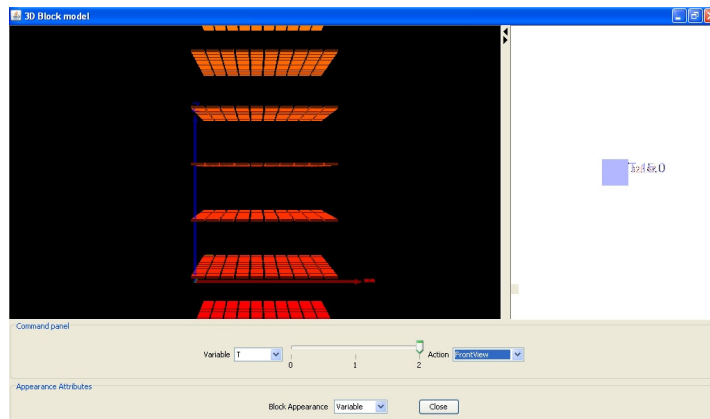


Fig. 17 - ExpandZ command result.

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 box (see Fig. 18).

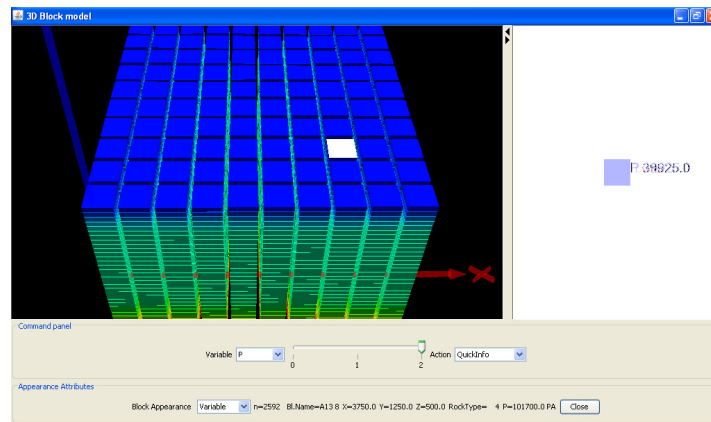


Fig. 18 - QuickInfo command result, information are printed in a panel at the bottom of the window.

Select the **Action->SetVOI** command to set the coordinates of a volume of interest. When the SetVOI dialog box 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 showed in Fig. 19.

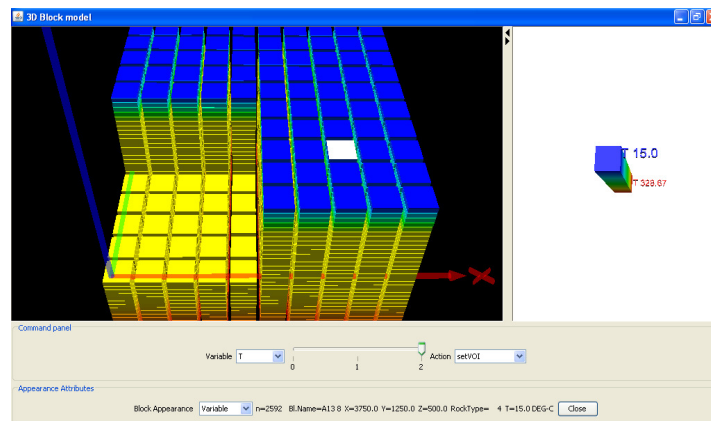


Fig. 19 - SetVOI function applied to the model.

From the **Block appearance** combo box, select **RockTypes**. The block scale colour is related to the rock type (as shown in Fig. 20).

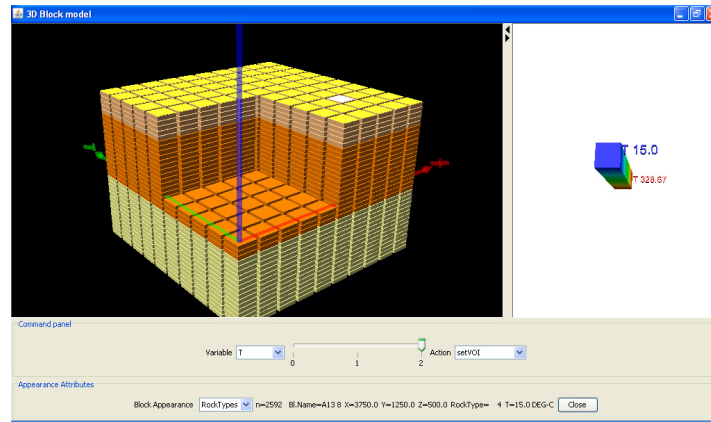



Fig. 20 - 3D Block model in material color scale.

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 box. From the main menu select **Tools->Option** to display the Option dialog box 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.

4.2.3 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 box is displayed as shown in Fig. 21.

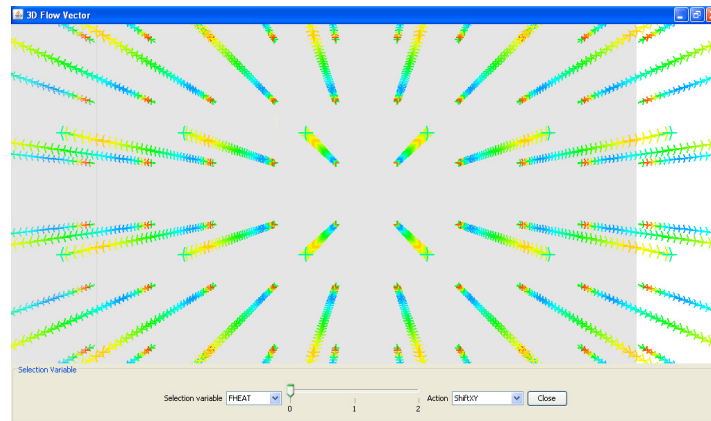


Fig. 21 - Vector representation of the heat flow.

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 box is displayed (see Fig. 22).

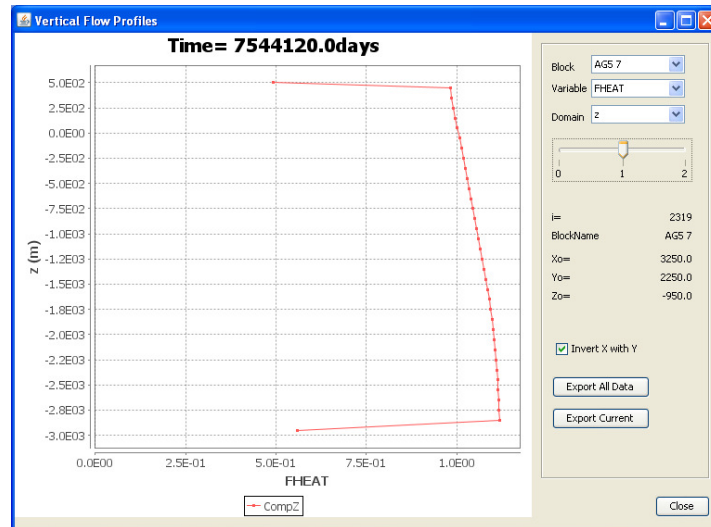



Fig. 22 - Vertical heat flow profile.

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.

4.2.4 3D Iso Surface

From TOUGH2Viewer menu bar, click **View->3D IsoSurface** (that is the three-dimensional analogue of the isoline), and then a **3D IsoSurface** dialog box is displayed as shown in Fig. 23.

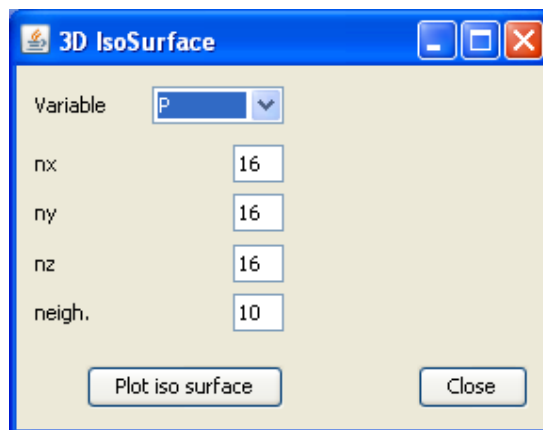


Fig. 23 - 3D IsoSurface dialog.

Click **Plot iso surface** button to obtain a **IsoSurface** dialog (see Fig. 24).

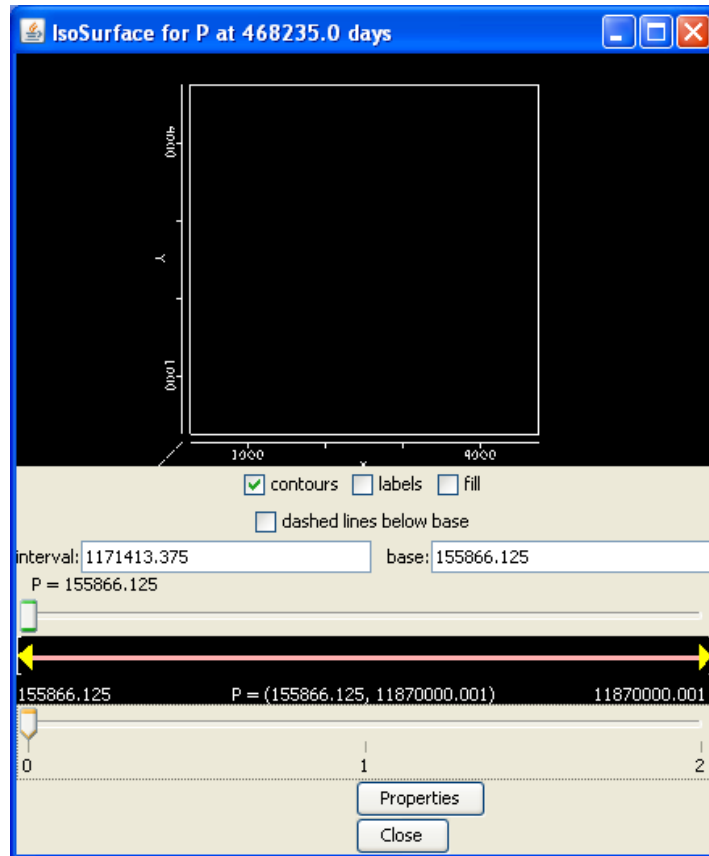


Fig. 24 - IsoSurface Dialog for pressure and time step 0.

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. 25.

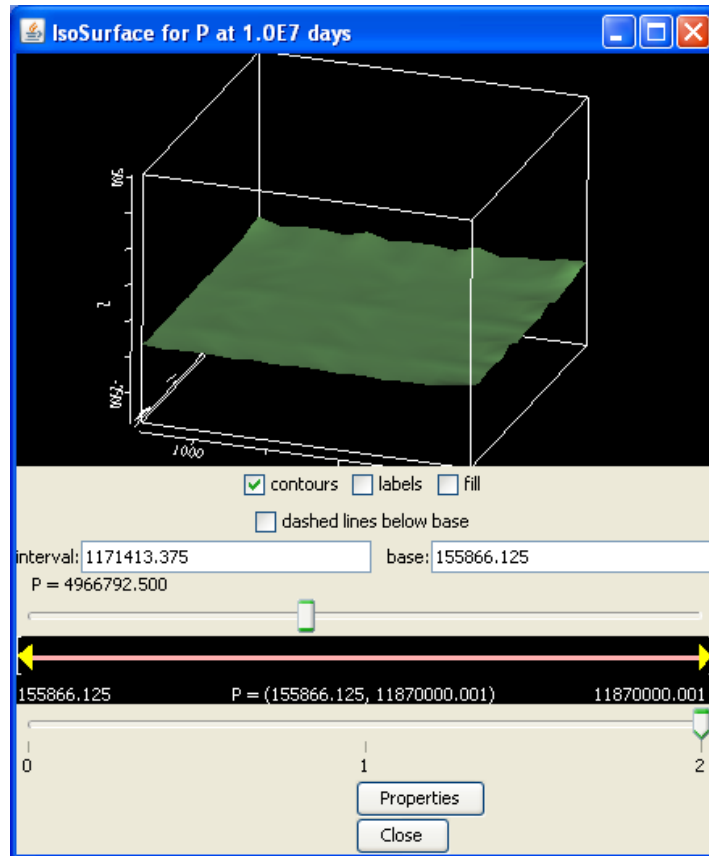




Fig. 25 - Dialog box for the pressure IsoSurface.

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.

4.2.5 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. 26.

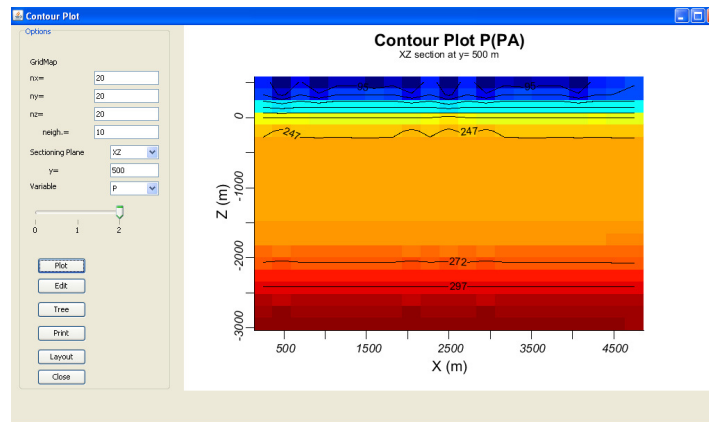




Fig. 26 - 2D Contour Plot for the Pressure (Pa).

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 box is displayed (see pag. 10).

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

4.3 Unstructured 3D grid model

This example is related to an unstructured grid model, built using a special GIS GRASS version adapted to create locally refined grids.

Start **TOUGH2Viewer** (Windows OS) by double clicking the batch file **go.bat** then the main window of TOUGH2Viewer is displayed, as shown in Fig. 27.

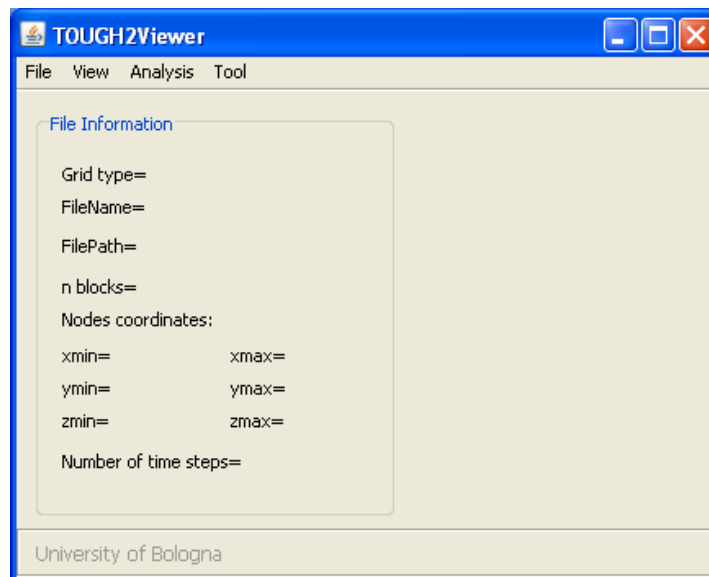


Fig. 27 - TOUGH2Viewer empty main window

4.3.1 Load Files

From main menu, click the **File->Open unstructured grid** menu item. The **Open unstructured grid** dialog box will be displayed (Fig. 28).

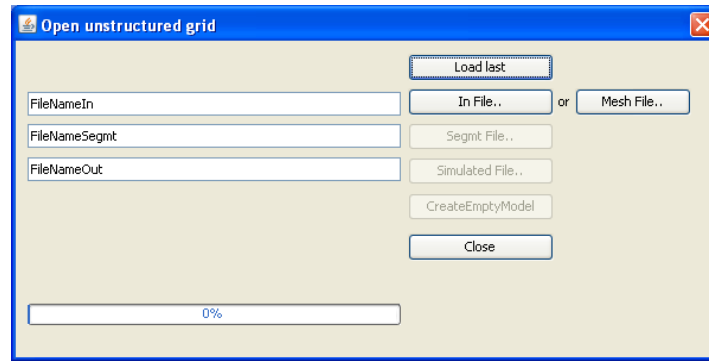


Fig. 28 - Open unstructured grid dialog box

Click the **In File** button to open the **Open In** dialog box 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 box 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 box 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. 29).

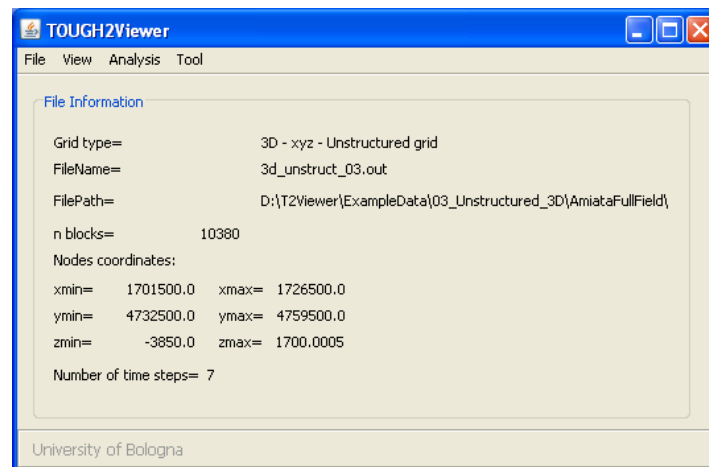


Fig. 29 - TOUGH2Viewer main window that summarizes some geometric data of the loaded numerical model.

In order 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 box is opened (Fig. 30)

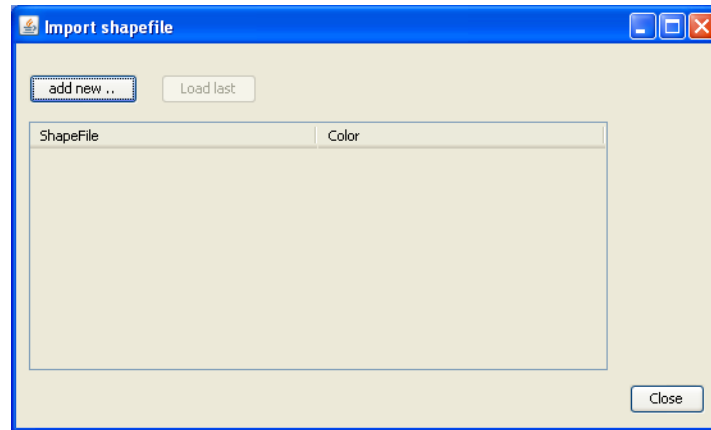


Fig. 30 - Import shapefile dialog Box.

Click the **add new ...** button to open the **LoadShapeFile** dialog box (Fig. 31).

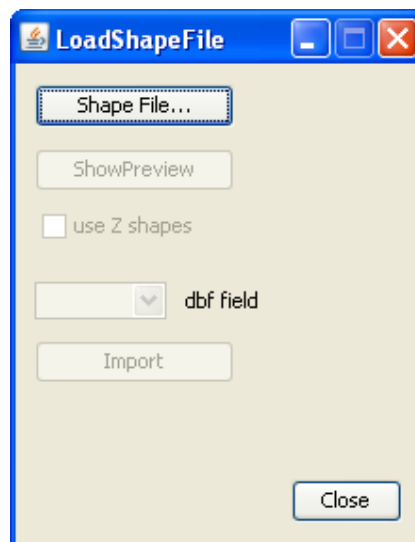


Fig. 31 - Dialog Box to load shape files.

Click the **Shape File...** button to display the **Open** dialog box and navigate to the **base_ss** folder. In particular go to `ExampleData\03_unstructured_3D\Maps\base_ss` folder and select the **“base_ss.shp”** file (Fig. 32).

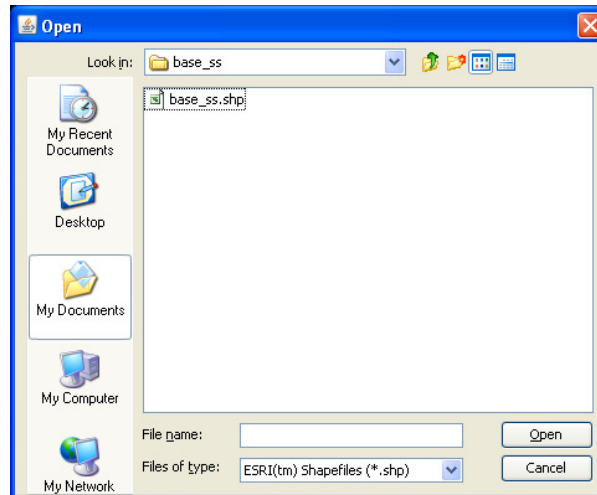


Fig. 32 - Dialog Box to select the file to be loaded.

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. 33).

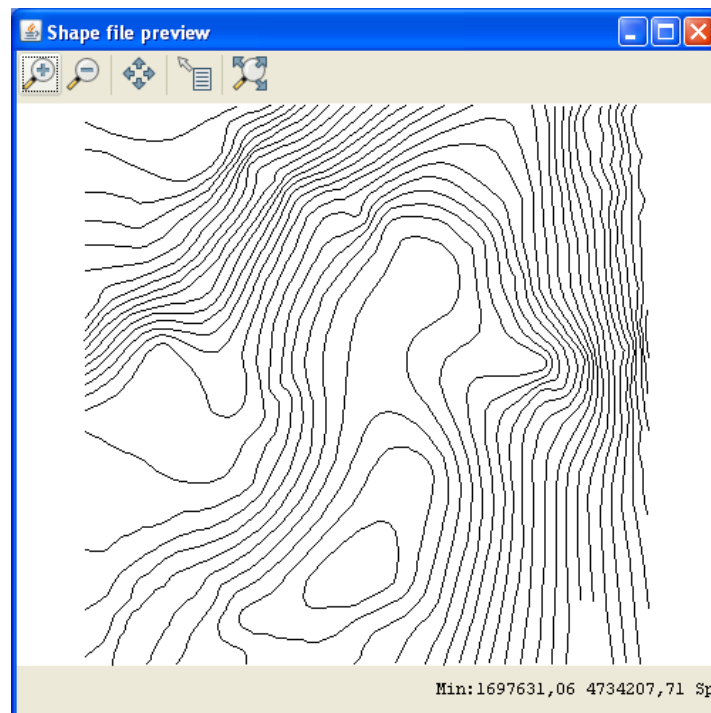



Fig. 33 - 2D Preview of the base_ss.shp file

Close the **Shape file preview** window by clicking the window icon . From the **LoadShapeFile** dialog box, select “**Quota**” item in the **dbf field** combo box (to load from a dbf file the elevation quote of the polylines object) (Fig. 34a) and click **Import** button (Fig. 34b).

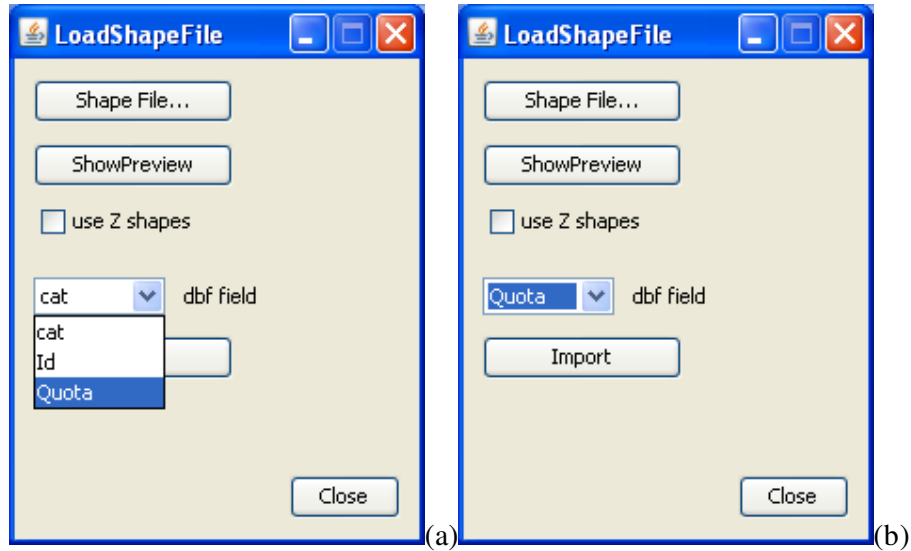


Fig. 34a, b - LoadShpFile Dialog window

The **Import shapefile** window now will be as showed in Fig. 35. The maximum number of loaded maps depends on the quantity of free RAM memory available.

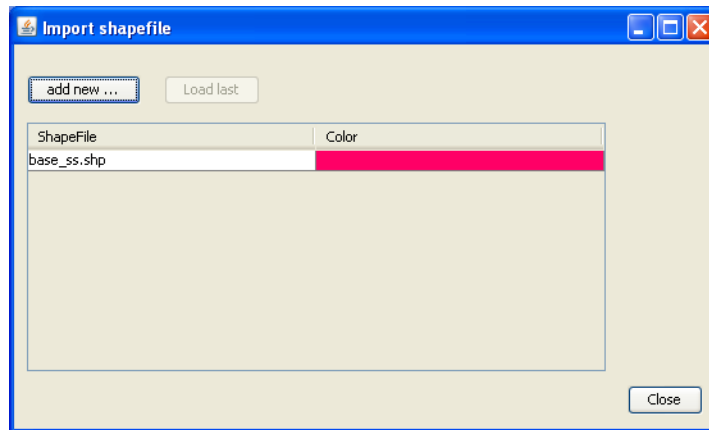



Fig. 35 - Import shapefile window. The table resume imported shape files.

By double clicking the **color** bar near the file name, the **Choose a color...** window dialog box is displayed to change the color of polylines (Fig. 36). 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.

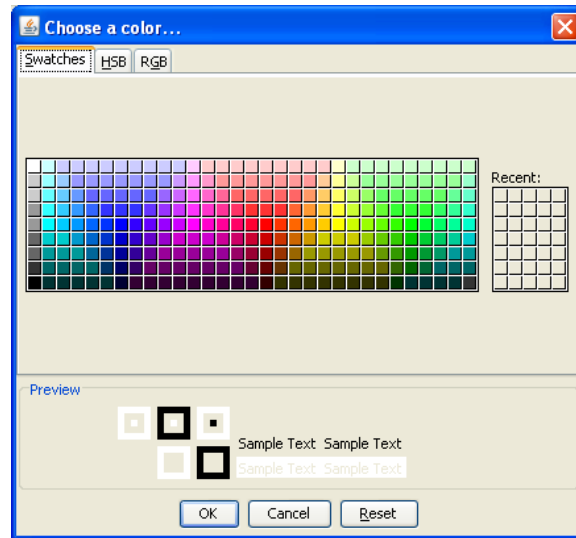


Fig. 36 – “Choose a color ...” dialog box window.

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

4.3.2 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. 37.

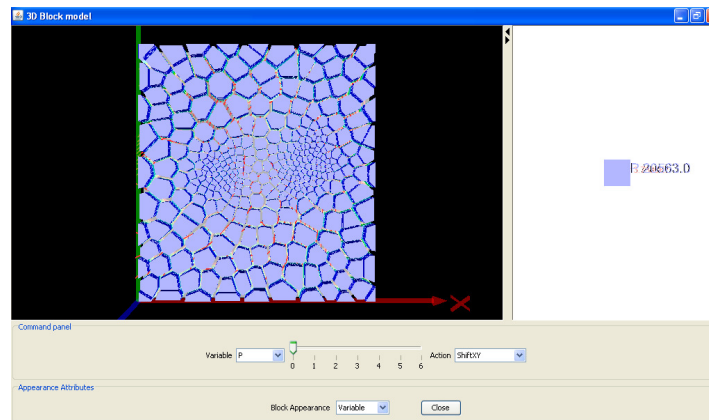


Fig. 37 - Top view of the 3D Block model.

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 box is displayed. In the panel **Hide VOI**, check the **Enable** checkbox, click both **Default** and **Apply Hide VOI** buttons (Fig. 38).

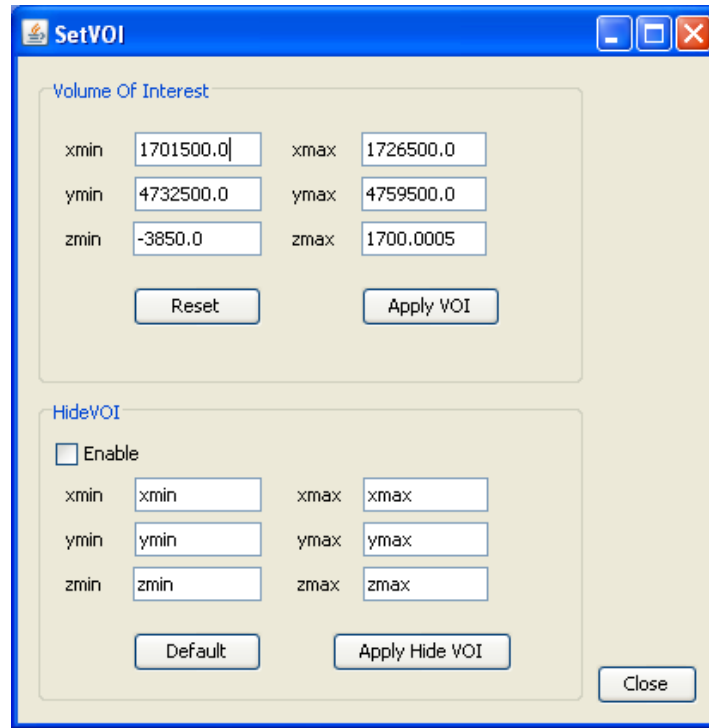



Fig. 38 - SetVOI dialog window.

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, 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. 39, (b)).

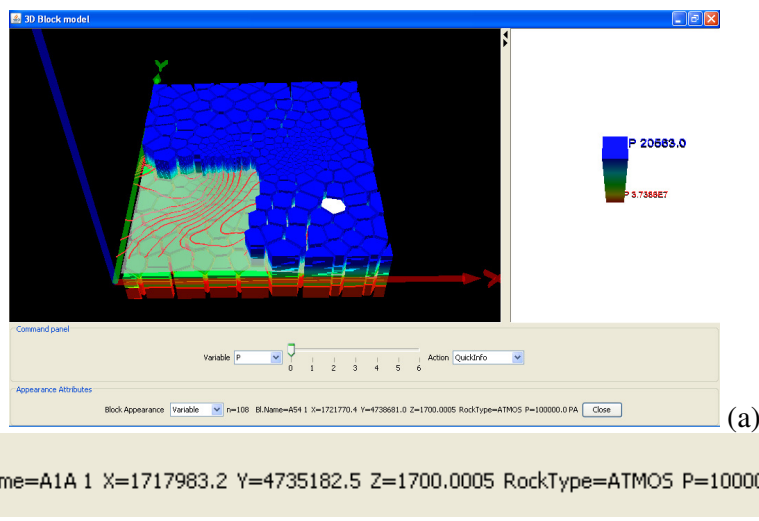



Fig. 39 - (a) 3D model partially showed and block selected (white block); (b) information related to the selected block.

From the **Action** combo box, select **FindBlocks** and the **Find Blocks** dialog box is displayed (Fig. 40). 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.

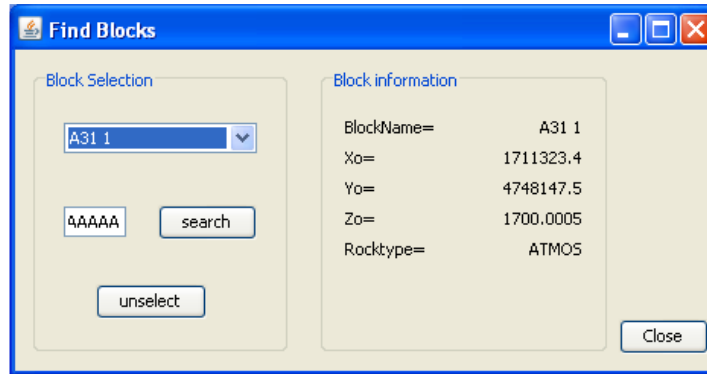



Fig. 40 - Find Block dialog box.

From the **Action** combo box, select **SnapShot** to display a **SnapShot Preview** dialog box that contains an image of the three-dimensional model (Fig. 41). 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 box by clicking the window icon  or via the **File->Exit** menu item.

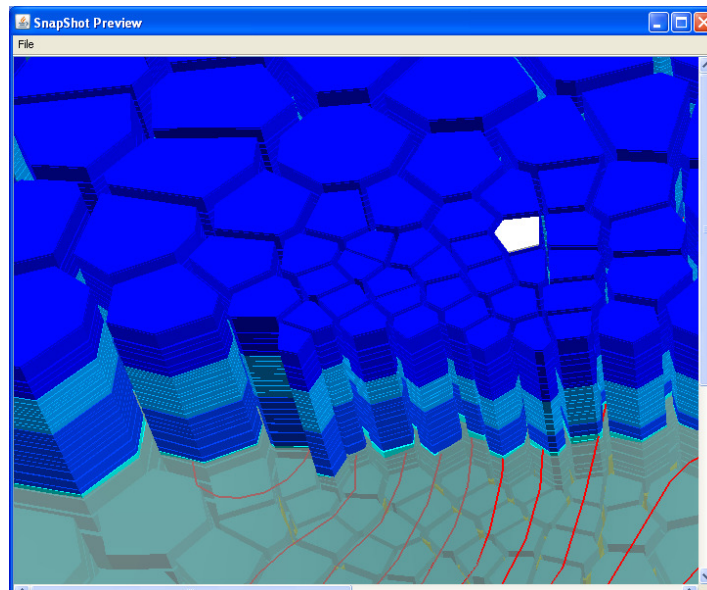


Fig. 41 - Snapshot Preview window dialog box.

To repaint each block in material scale color (Fig. 42), from the **Action** combo box of the **3D Block model** dialog box, execute the following commands:

- **ResetPositions;**
- **FrontView;**

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

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

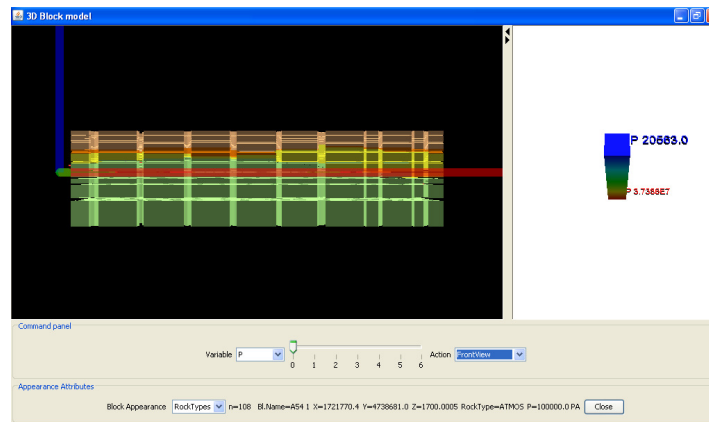


Fig. 42 - 3D Block Model front view. Material color scale.

From main menu, choose **Tools-> Options**, then click **3D Block model object** tab to open the Options dialog box (Fig. 43).

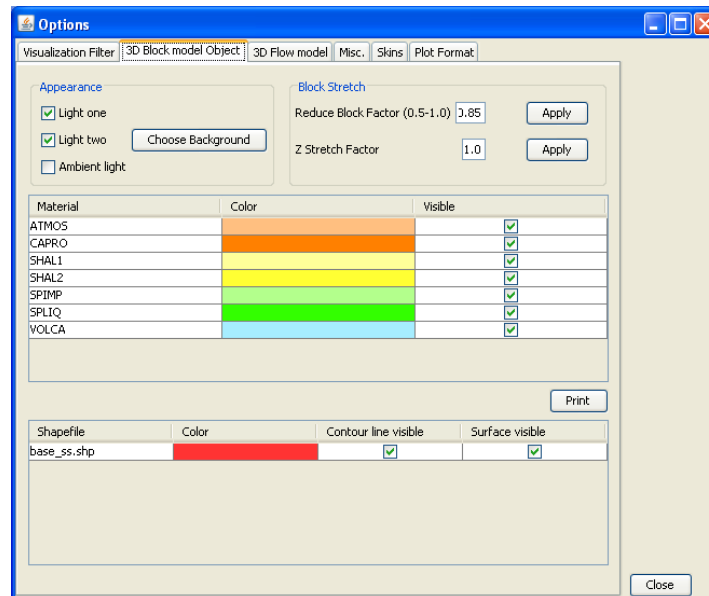


Fig. 43 - Options dialog window.

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. 44. Only the materials SPIMP and SPLIQ are kept visible.

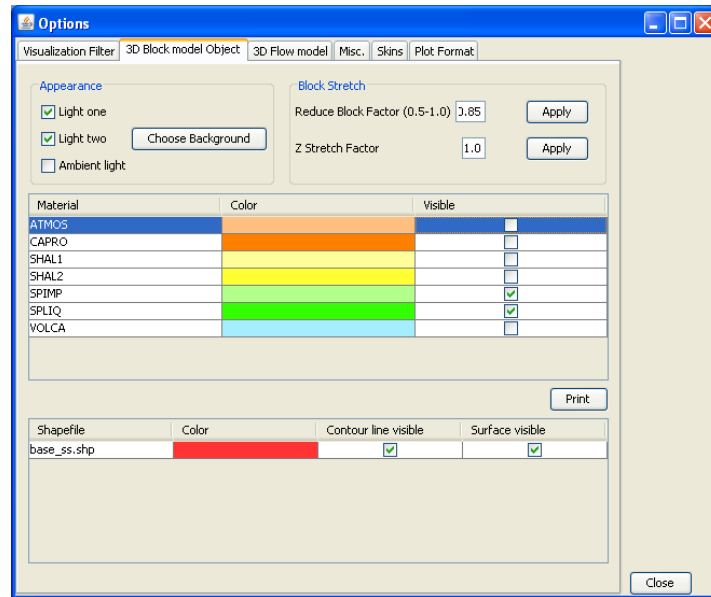



Fig. 44 - Options dialog with modified parameters.

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. 45.

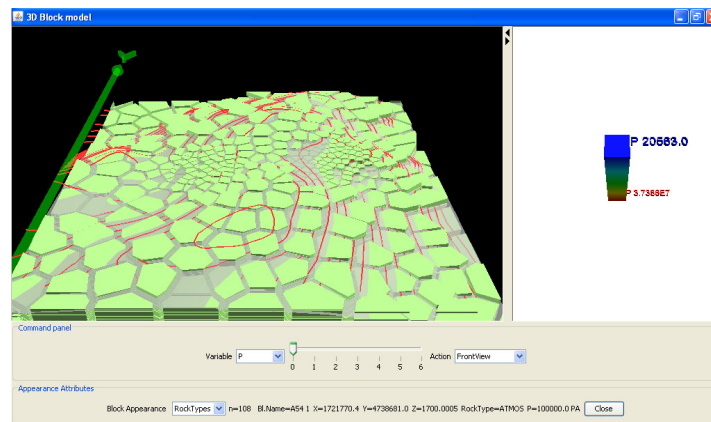


Fig. 45 - 3D Block model.

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 box is displayed.

From the slider menu select the time step “6”. The graph will be update as shown in Fig. 46.

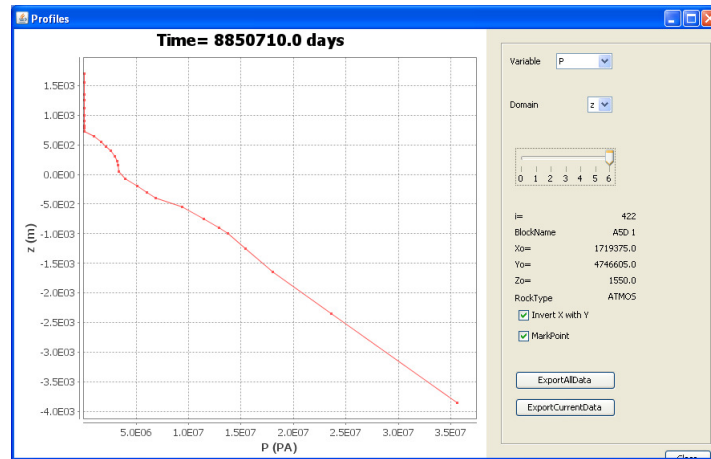



Fig. 46 - Profiles dialog which shows the pressure values along z axis.

Close the **Profiles** dialog box by clicking the  windows icon or the **Close** button.

In the **Options** dialog box, 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. 47.

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).

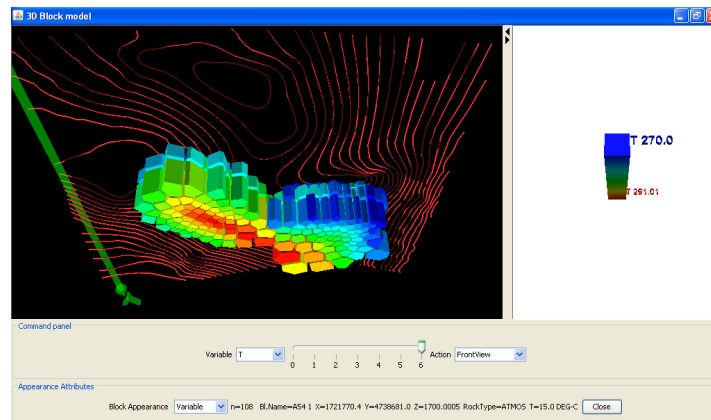



Fig. 47 - Bottom filtered view of the 3D model with modified color scale.

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

4.3.3 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. 48.

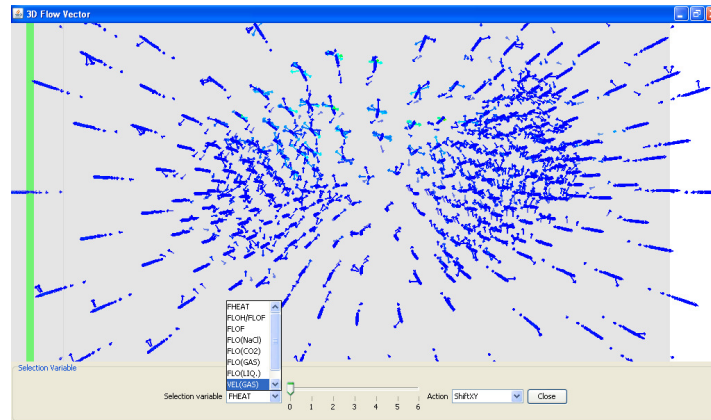


Fig. 48 - 3D Flow vector default viewing.

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 desired position (Fig. 49).

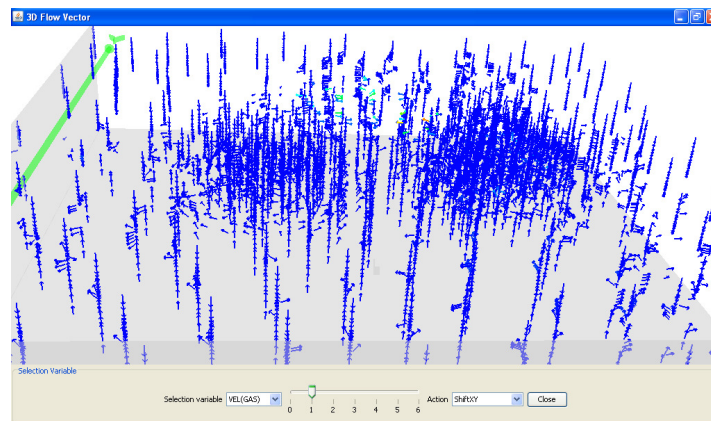




Fig. 49 - 3D Flow model at the time step 1 for the variable VEL(GAS).

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.

4.3.4 3D Iso Surface

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

Open a **3D Iso Surface** dialog (Fig. 50) 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.

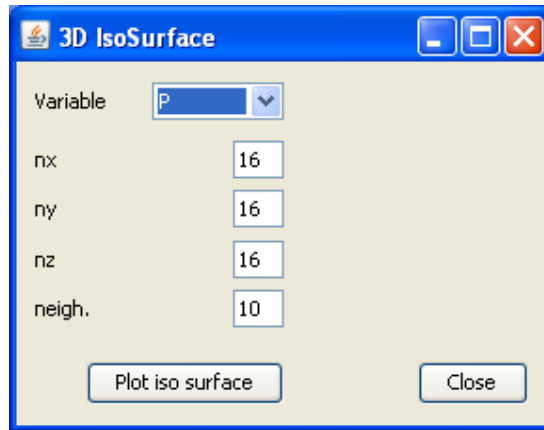


Fig. 50 - 3D IsoSurface dialog box.

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.

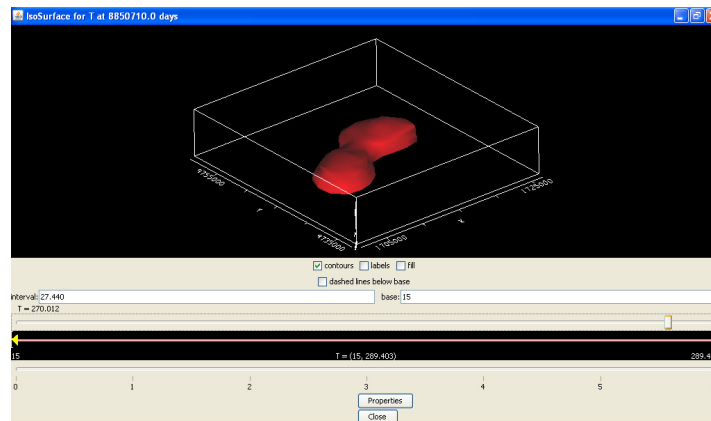


Fig. 51 - Isosurface for T=270°C, time step “6”.

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

4.3.5 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. 52.

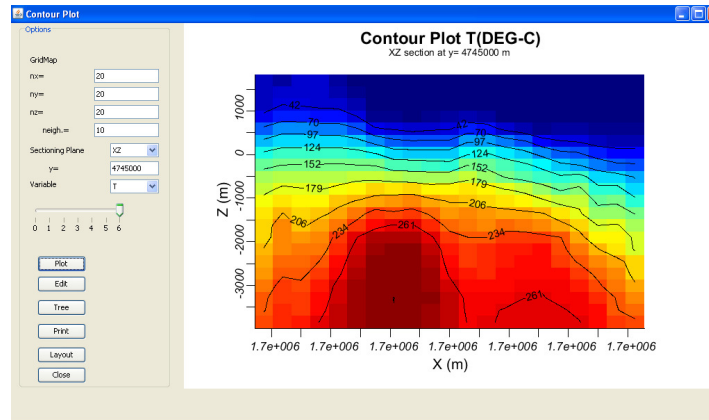




Fig. 52 – Temperature isolines contour map.

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.

4.3.6 Statistics

Click the **Analysis-> Statistics** menu item. The **Statistics** dialog box is displayed (see pag. 10.).

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

5 List of figures

Fig. 1 - The TOUGH2Viewer empty main window.....	6
Fig. 2 - The “Open structured grid” dialog box.....	6
Fig. 3 - Main window with geometric information of the loaded grid.	7
Fig. 4 - 3D Default representation of the Model, Top View.	7
Fig. 5 - 3D representation of the model, FrontView.	8
Fig. 6 - Pressure contour map.....	8
Fig. 7 - Temperature contour map.....	9
Fig. 8 - Vector map of the heat flow.	9
Fig. 9 - Vector map of the variable VEL(LIQ.).....	10
Fig. 10 - Min and Max value for simulation data. The default plot is first variable vs time.	10
Fig. 11 - Open Structured grid dialog.	11
Fig. 12 - 3D Block model window, top view.	11
Fig. 13 - 3D Block model box showing the Temperature variable at the time step 2.	12
Fig. 14 - 3D modelFrame horizontally sectioned.	12
Fig. 15 – Graph of the pressure versus the x coordinate.	13
Fig. 16 – Graph of the pressure versus the z coordinate.	13
Fig. 17 - ExpandZ command result.....	13
Fig. 18 - QuickInfo command result, information are printed in a panel at the bottom of the window.	14
Fig. 19 - SetVOI function applied to the model.	14
Fig. 20 - 3D Block model in material color scale.....	15
Fig. 21 - Vector representation of the heat flow.....	15
Fig. 22 - Vertical heat flow profile.	16
Fig. 23 - 3D IsoSurface dialog.....	16
Fig. 24 - IsoSurface Dialog for pressure and time step 0.....	17
Fig. 25 - Dialog box for the pressure IsoSurface.....	18
Fig. 26 - 2D Contour Plot for the Pressure (Pa).	19
Fig. 27 - TOUGH2Viewer empty main window.....	19
Fig. 28 - Open unstructured grid dialog box.....	20
Fig. 29 - TOUGH2Viewer main window that summarizes some geometric data of the loaded numerical model.	20
Fig. 30 - Import shapefile dialog Box.	21
Fig. 31 - Dialog Box to load shape files.....	21
Fig. 32 - Dialog Box to select the file to be loaded.	22
Fig. 33 - 2D Preview of the base_ss.shp file.....	22
Fig. 34a, b - LoadShpFile Dialog window.....	23
Fig. 35 - Import shapefile window. The table resume imported shape files.	23
Fig. 36 – “Choose a color ...” dialog box window.....	24
Fig. 37 - Top view of the 3D Block model.	24
Fig. 38 - SetVOI dialog window.....	25
Fig. 39 - (a) 3D model partially showed and block selected; (b) information related to the selected block.....	25
Fig. 40 - Find Block dialog window.	26
Fig. 41 - Snapshot Preview window dialog box.	26
Fig. 42 - 3D Block Model front view. Material color scale.	27
Fig. 43 - Options dialog window.	27
Fig. 44 - Options dialog with modified parameters.	28
Fig. 45 - 3D Block model.....	28
Fig. 46 - Profiles dialog which shows the pressure values along z axis.	29

Fig. 47 - Bottom filtered view of the 3D model with modified color scale.	29
Fig. 48 - 3D Flow vector default viewing.	30
Fig. 49 - 3D Flow model at the time step 1 for the variable VEL(GAS).	30
Fig. 50 - 3D IsoSurface dialog box.	31
Fig. 51 - Isosurface for T=270°C, time step "6"	31
Fig. 52 - Temperature isolines contour map.	32