User Defined Geometry

Workflow for MATLAB non-specialist users

Working document!!!

Introduction

This slide pack illustrates how to run a new model geometry in mine_geothermal without having to do any significant amount of programming.

It is aimed at users who may have a basic familiarisation with MATLAB, but are not confident/experienced in adding their own code to the program files.

This workflow assumes that the user has access to the Mapping Toolbox in their distribution of MATLAB

It also assumes that the relevant .shp files have been prepared prior to running mine_geothermal

TODO - add info on setting path etc.

Setting Physical Properties

You have two options.

1) Run `mine_geothermal` with default rock and fluid properties
2) Run `mine_geothermal` with user specified rock and fluid properties

USE DEFAULT PROPERTIES FOR NOW!
SLIDES NOT COMPLETE FOR USER OPTIONS

Option 1: To use default physical properties, set
`physical_propertiesFlag = 2`

```
45        % Material physical properties options
46 -      physical_propertiesFlag = 2;      %%% 0 - User specified + command prompt
47                                          %%% 1 - User specified - define in code
48                                          %%% 2 - Default properties
```

Setting Physical Properties

Option 2: TODO - Add text on how to create a material properties file The file is read in using `readtable`. This creates a MATLAB object. The attributes of this object are the column headings. So long as you have correctly named your headings, the specific order doesn't matter.
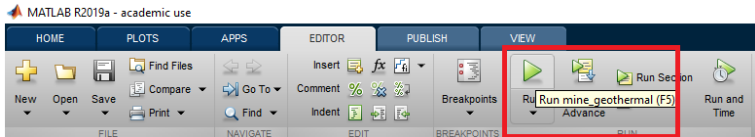
Beginning Model Set-up

Open `mine_geothermal.m`

Set `igeom` = `'UserDefinedGeometry-CommandLinePrompt'`

Run `minegeothermal.m`

A series of pop-up windows will now appear.

Case Selection

The first pop-up will ask you to how many seams you wish to model.
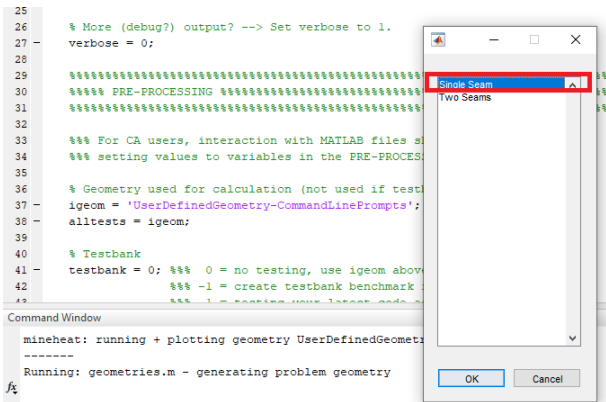
Select "Single Seam" and click "OK"

File Selection

File selection prompt

The next pop-up will ask you to select a shapefile (.shp).

```
23
24
25
26      % Geometry used for calculation (not used i
27 -    igeom = 'UserDefinedGeometry-CommandLineP
28 -    alltests = igeom;
29
30      % More (debug?) output? --> Set verbose to 1
31 -    verbose = 0;
```

Select shapefile (.shp)

OK

Click "OK". This will open a file selection window.

File Selection

Navigation window

Navigate to your desired shapefile
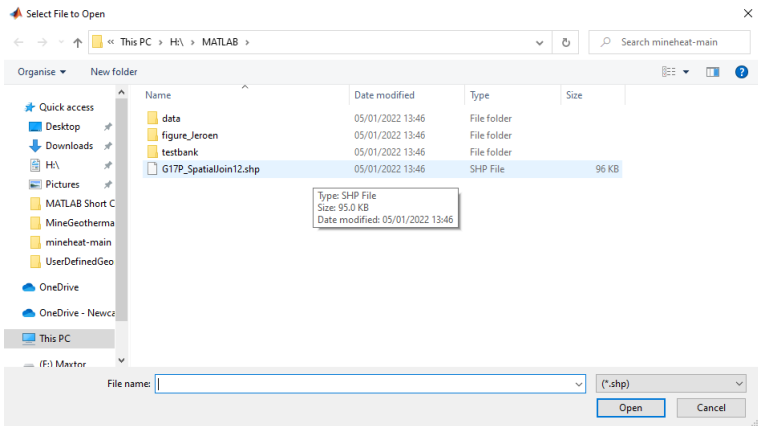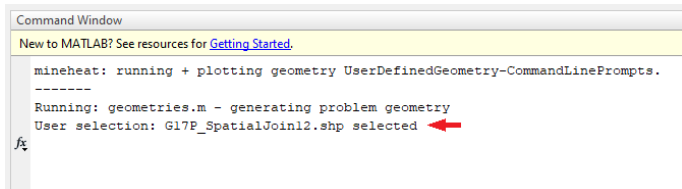
Click "Open"

File Selection

Successful selection

If selection is successful, a message to this effect will print to the command line

Inflow/Outflow Selection

Once the shapefile has been successfully selected, a second pop-up will appear asking you to specify the number of inflows and outflows you wish to apply to the model.
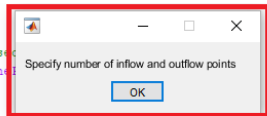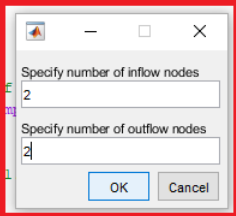

Click "OK". An input box will then appear

Inflow/Outflow Selection

Number of inflow/outflow nodes


```
20    % 20210609 - added igeom 101 and 102 to test prescribed in/outflow
21    % 20210518 - merging different codes into a signle master version
22    % 20190628 - code split up in separate subfunctions
23    %
24    % Jeroen van Hunen
25
26    % Geometry used for calculation (not used if
27 -  igeom = 'UserDefinedGeometry-CommandLinePro
28 -  alltests = igeom;
29
30    % More (debug?) output? --> Set verbose to 1
31 -  verbose = 0;
32
```

Dialog box:
Specify number of inflow nodes
2
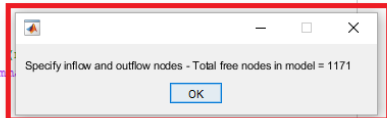Specify number of outflow nodes
2
OK   Cancel

For now, we will specify 2 inflow and 2 outflow points. Click "OK".

Inflow/Outflow Selection

Another pop-up box will appear asking you to specify the node numbers at which you wish to specify inflow and outflow.

Note: The pop-up box will tell you how many free nodes are available in your model.
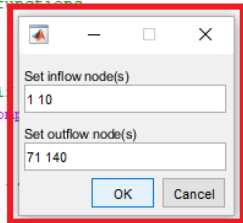

Click "OK". An input box will now appear.

Inflow/Outflow Selection

Inflow/outflow node numbers

Specify the inflow and outflow **node numbers** as space-separated lists. Here we specify nodes **1** and **10** as **inflow**, and nodes **71** and **140** as **outflow**.

Click "OK".

Note: Nodes don't need to be specified in a particular numerical order.

Inflow/Outflow Selection
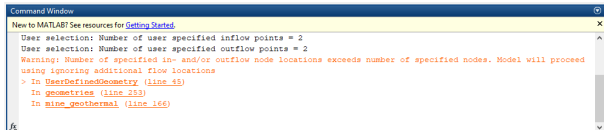
If **too few** nodes are specified the following `Warning:` will be returned to the command line. You will then be asked to re-specify node numbers.

If **too many** nodes are specified in the then additional entries are ignored. A `Warning:` is printed to the command line, and the code continues.
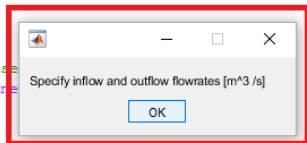
Flow Rates

A pop-up will now ask you to specify the **flow rates** at each node.

Note: `MineGeothermal` requires flows to be specified as absolute values for inflow and outflow.

```
21     % 20210518 - merging different codes into a signle master version
22     % 20190628 - code split up in separate subfunctions
23     %
24     % Jeroen van Hunen
25
26     % Geometry used for calculation (not use
27 -    igeom = 'UserDefinedGeometry-CommandLine
28 -    alltests = igeom;
29
30     % More (debug?) output? --> Set verbose to 1.
```
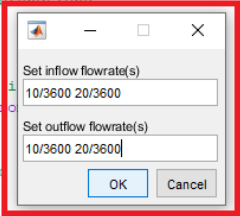

Specify inflow and outflow flowrates [m^3 /s]

OK

Click "OK". An input box will now appear.

Flow Rates

Specify flow rates

Specify flow rates as a **space-separated list**. You need to specify flow rates in m^3/s^{-1}.

```
21    % 20210518 - merging different codes into a single master version
22    % 20190628 - code split up in separate subfunctions
23    %
24    % Jeroen van Hunen
25
26    % Geometry used for calculation (not used i
27 -  igeom = 'UserDefinedGeometry-CommandLinePi
28 -  alltests = igeom;
29
30    % More (debug?) output? --> Set verbose t
31 -  verbose = 0;
32
```

Set inflow flowrate(s)
```
10/3600 20/3600
```

Set outflow flowrate(s)
```
10/3600 20/3600
```

OK   Cancel

You should specify flow rates in the same order you specified nodes in the previous step. e.g.
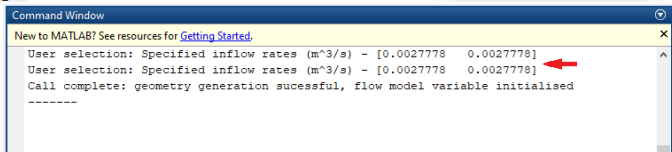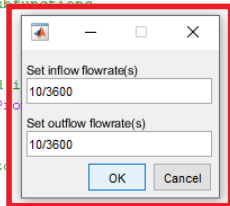
Node **10** has an inflow rate of $20/3600$ m^3/s^{-1}.

Node **71** has an outflow rate of $10/3600$ m^3/s^{-1}.

Flow Rates

Special cases: All wells pumped at the same rate

If all of your inflow and/or outflow wells have the same flow rate you only need to specify one value. This flow rate will be automatically assigned to the remaining nodes.

Flow Rates

Special cases: Uniform inflow rates and variable outflow rates

We have 2 inflow and 2 outflow wells. The **total** inflow and outflow rates are both 30 m^3/hour.
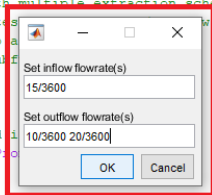
This is split evenly between both inflow wells:
      15 m^3/hour for each
There is an uneven distribution over the outflow wells
      Well A: 10 m^3/hour   Well B: 20 m^3/hour.

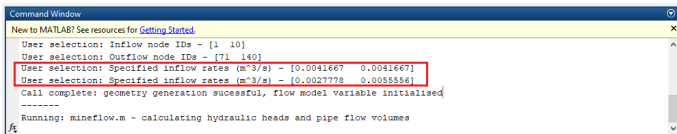This is implemented as follows:

```
17   %           put testbank procedures in separate function testbank_eval
18   % 20210630 - added testbank
19   % 20210628 - added igeom 103 for grid with multiple extraction schemes
20   % 20210609 - added igeom 101 and 102 to te
21   % 20210518 - merging different codes into a
22   % 20190628 - code split up in separate su
23   %
24   % Jeroen van Hunen
25
26   % Geometry used for calculation (not used
27 - igeom = 'UserDefinedGeometry-CommandLineP
28 - alltests = igeom;
29
```

Set inflow flowrate(s)
`15/3600`

Set outflow flowrate(s)
`10/3600 20/3600`

[OK] [Cancel]

Flow Rates

Special cases: Uniform inflow rates and variable outflow rates

Click "OK", and the following will display on the command line.

See that both inflow wells have been assigned the same flow rate, and the differeing outflow rates have also been assigned.

Note: The opposite case with uniform outflow rates and variable inflow rates is equally possible and specified in the same manner as this example.

Once flow rates have been set, the model will run.