

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`

Making mine_geothermal Available to MATLAB

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.

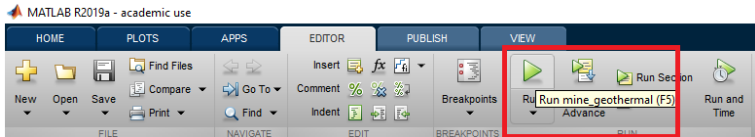
Beginning Model Set-up

Open mine_geothermal.m

Set igeom = 'UserDefinedGeometry-CommandLinePrompt'

```
25  
26 % Geometry used for calculation (not used if testbank!=0):  
27 - igeom = 'UserDefinedGeometry-CommandLinePrompts';  
28 - alltests = igeom;  
29
```

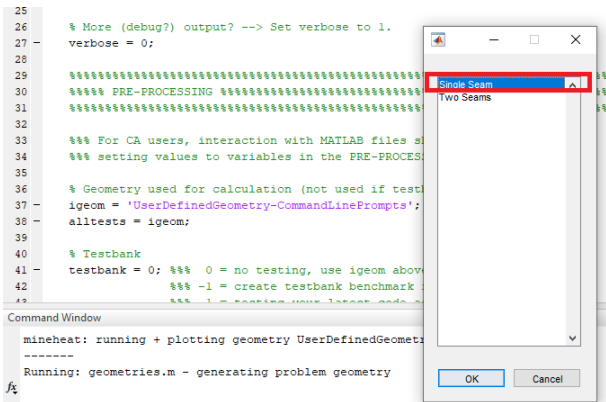
Run minegeothermal.m



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.



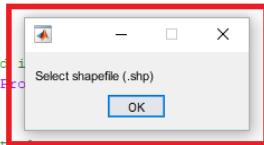
Select “Single Seam” and click “OK”

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-CommandLinePro
28 - alltests = igeom;
29
30 % More (debug?) output? --> Set verbose to 1.
31 - verbose = 0;
```

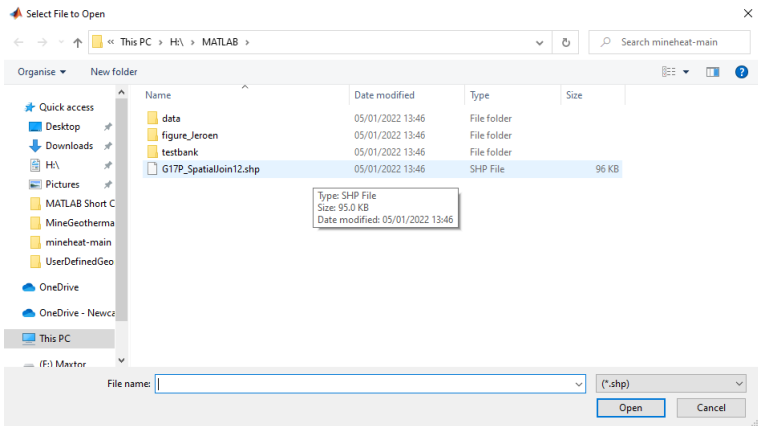


Click “OK”. This will open a file selection window.

File Selection

Navigation window

Navigate to your desired shapefile

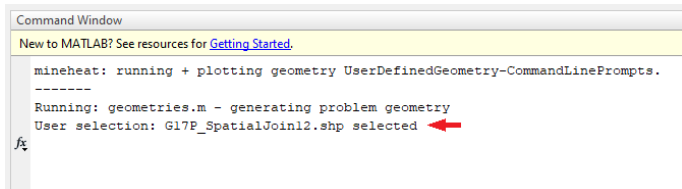


Click “Open”

File Selection

Successful selection

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



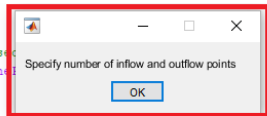
A screenshot of the MATLAB Command Window. The title bar reads "Command Window". Below the title bar is a yellow banner with the text "New to MATLAB? See resources for [Getting Started.](#)". The command window contains the following text: "mineheat: running + plotting geometry UserDefinedGeometry-CommandLinePrompts.", followed by a line of dashes "-----", then "Running: geometries.m - generating problem geometry", and finally "User selection: G17P_SpatialJoin12.shp selected" with a red arrow pointing to the end of the line. On the left side of the command window, there is a vertical toolbar with a "fx" icon.

```
Command Window
New to MATLAB? See resources for Getting Started.
mineheat: running + plotting geometry UserDefinedGeometry-CommandLinePrompts.
-----
Running: geometries.m - generating problem geometry
User selection: G17P_SpatialJoin12.shp selected
```

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.

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

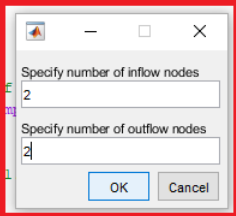


Click “OK”. An input box will then appear

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 single 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-CommandLineProm
28 - alltests = igeom;
29
30 % More (debug?) output? --> Set verbose to 1
31 - verbose = 0;
32
```



Specify number of inflow nodes

2

Specify number of outflow nodes

2

OK Cancel

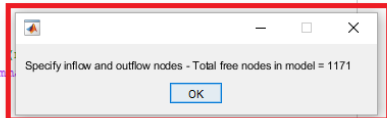
For now, we will specify 2 inflow and 2 outflow points. Click “OK”.

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.

```
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
27 - igeom = 'UserDefinedGeometry-Com
28 - alltests = igeom;
29
30 % More (debug?) output? --> Set verbose to 1.
```



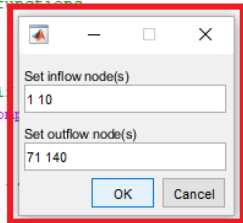
Click "OK". An input box will now appear.

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**.

```
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 in
27 - iggeom = 'UserDefinedGeometry-CommandLineProgra
28 - alltests = iggeom;
29
30 % More (debug?) output? --> Set verbose to 1
31 - verbose = 0;
32
```



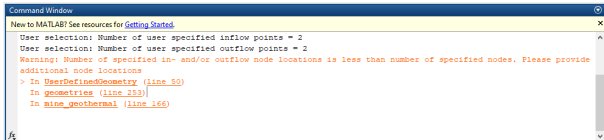
Click “OK”.

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

Inflow/Outflow Selection

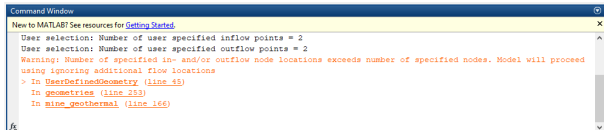
Warnings: Too many/too few points

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.



```
Command Window
New to MATLAB? See resources for Getting Started.
User selection: Number of user specified inflow points = 2
User selection: Number of user specified outflow points = 2
Warning: Number of specified in- and/or outflow node locations is less than number of specified nodes. Please provide additional node locations
> In UserDefinedGeometry (line 50)
   In geometries (line 253)
   In mine_geothermal (line 166)
```

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.



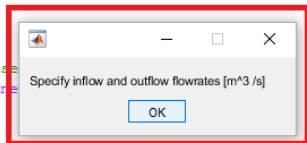
```
Command Window
New to MATLAB? See resources for Getting Started.
User selection: Number of user specified inflow points = 2
User selection: Number of user specified outflow points = 2
Warning: Number of specified in- and/or outflow node locations exceeds number of specified nodes. Model will proceed using ignoring additional flow locations
> In UserDefinedGeometry (line 45)
   In geometries (line 253)
   In mine_geothermal (line 166)
```

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



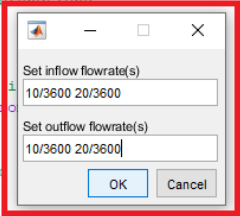
Click “OK”. An input box will now appear.

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 in
27 - igeom = 'UserDefinedGeometry-CommandLinePro
28 - alltests = igeom;
29
30 % More (debug?) output? --> Set verbose to
31 - verbose = 0;
32
```



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 $20/3600 \text{ m}^3/\text{s}^{-1}$.

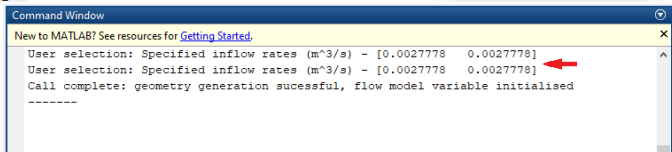
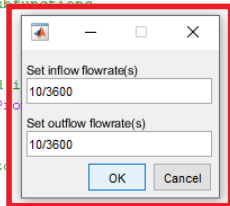
Node **71** has an outflow rate of $10/3600 \text{ m}^3/\text{s}^{-1}$.

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.

```
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-CommandLineP
28 - alltests = igeom;
29
30 % More (debug?) output? --> Set verbose to
31 - verbose = 0;
32
```



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 m³/hour.

This is split evenly between both inflow wells:

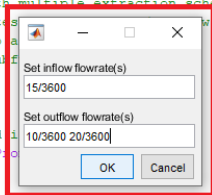
15 m³/hour for each

There is an uneven distribution over the outflow wells

Well A: 10 m³/hour Well B: 20 m³/hour.

This is implemented as follows:

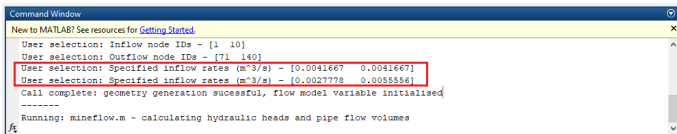
```
17 % put testbank procedures in separate function testbank_eval
18 % 20210630 - added testbank
19 % 20210628 - added igeom 103 for grid with multiple extraction schemes
20 % 20210609 - added igeom 101 and 102 to testbank
21 % 20210518 - merging different codes into a single function
22 % 20190628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



Flow Rates

Special cases: Uniform inflow rates and variable outflow rates

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



```
Command Window
New to MATLAB? See resources for Getting Started.
User selection: Inflow node IDs - [1 10]
User selection: Outflow node IDs - [71 140]
User selection: Specified inflow rates (m^3/s) - [0.0041667 0.0041667]
User selection: Specified inflow rates (m^3/s) - [0.0027778 0.0055556]
Call complete: geometry generation successful, flow model variable initialised
-----
Running: mineflow.m - calculating hydraulic heads and pipe flow volumes
```

See that both inflow wells have been assigned the same flow rate, and the differing 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.

Calculate

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