

# **Y-Mat Code**

## **A brief User Guide**

November 2018

**Gang Liu, Jiayuan Cao**

Key Laboratory of Shale Gas and Geoengineering,  
Institute of Geology and Geophysics,  
Chinese Academy of Sciences,  
No. 19, Beitucheng Western Road, Chaoyang District, 100029, Beijing, P.R.China  
E-mail address: [liugang\\_iggcas@163.com](mailto:liugang_iggcas@163.com) (G. Liu)

### **Table of Contents**

<b>1 Introduction.....</b>	<b>2</b>
<b>2 Pre-processing settings .....</b>	<b>3</b>
2.1 Data preparation .....	3
2.2 Data read using MATLAB .....	3
<b>3 Running settings .....</b>	<b>4</b>
3.1 Running option .....	4
3.2 Material assignment.....	4
3.3 Boundary conditions.....	5
3.4 Mechanical calculation .....	6
<b>4 Post-processing in MATLAB.....</b>	<b>6</b>
4.1 Results of the failure trajectories and modes .....	7
4.2 Results of the displacement of the model .....	8
4.3 Results of the stress distribution of the model .....	8

## 1 Introduction

This manual provides a brief usage of Y-Mat code, taking Brazilian disc test as an example. The code should be run using MATLAB software. For better understanding the source code, functions and script files are described as follows:

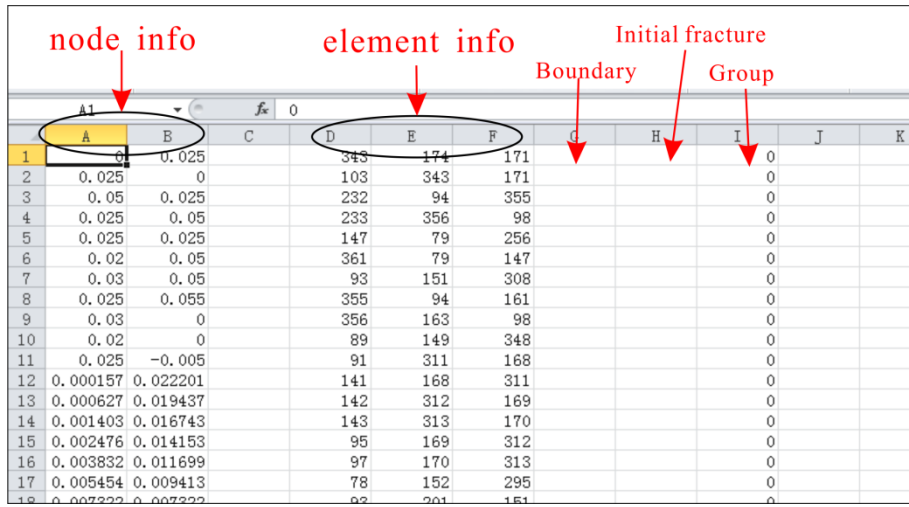
Name of the function or script file	Brief description of the function or script file
readata.m	Read the raw grid data generated by other pre-processing software like gmsh.exe
Moget1NewPoint1.m	Prepare for the information of nodes, elements, boundary nodes, DFN nodes, and groups.
get2arean.m	Calculate element area via the element's id
get2Ecentern.m	Calculate the coordinates of the center point of all the elements
get2lengn.m	Calculate length of an edge of element
moint2.m	Get the information of interface between two elements (Boundary interface or DFN)
get3arean.m	Calculate element area via the coordinates of the element
get3outnorm.m	Calculate the external normal vector of an edge of element
getmaxr.m	Calculate the data for NBS method
get4rrn.m	Calculate data for NBS method
mogetmass.m	Calculate the mass matrix of all the elements
modamp.m	Calculate viscous damping coefficient vector
Young2Shear.m	Transform Young modulus $E$ and poisson ratio $\nu$ into bulk modulus and shear modulus
calculate.m	Calculate the contact force of the discrete elements
inoutriangle.m	Judge the relative positional relationship between point and triangle
line2line.m	Determine if there are intersections between the two segments
line2triangle.m	Achieve the intersections between segment and triangle
moNBSmodify.m	Modify the contact information during running using Munjiza No Binary Search contact detection algorithm
mo fracture.m	Calculate the joint element force including crack model and DFN
fst_cal.m	Calculate the initial shear strength matrix of the elements
moinitialcontact.m	Get the initial contact information of the crack model and DFN
sigma_cal.m	Calculate the deformation forces matrix of elements
chplpr.m	Calculate the elastic matrix according to the elastic modulus $E$ and Poisson's ratio $\nu$
mo frd.m	Correction of elements large deformation
mogetboundaryU.m	Set the stress boundary conditions or initialize the displacement boundary conditions
modifyVol1.m	Set the displacement boundary conditions
initialize1.m	Initialize the essential vectors/ matrixes used in the code
groupinter.m	Input the mechanics parameters of crack model
groupzone.m	Input mechanics parameters of elements
run1.m	The main program
stepsize.m	Calculate the time step size using Young modulus $E$ , poisson ratio $\nu$ and density $\rho$
getimagen.m	Plot the model using current coordinate and fractures
plot_dis.m	Plot the result of displacement (eg. x-component, y-component and the total displacement)
plot_sig.m	Plot the result of stress (eg. x-component, y-component, shear stress, min principal stress, max principal stress)

## 2 Pre-processing settings

### 2.1 Data preparation

The grid used in Y-Mat code can be obtained by many meshing softwares such as Gid, Gmsh, et al. No matter which softwares are used, the data should include the information of nodes, elements, boundary nodes, fracture nodes and groups. Currently, the grids are generated using Gmsh (a meshing software).

The data are saved in an excel file, as shown in Fig.1. And each column represent different information of data. The data in columns A and B are the x and y coordinates of the nodes, respectively. The data in columns D, E and F are the elements information (three nodes' id of an element). The data in column G are the boundary node information. The data in column H are the initial fracture node information. The data in column I are the group information. The order of the column information should be strictly enforced.

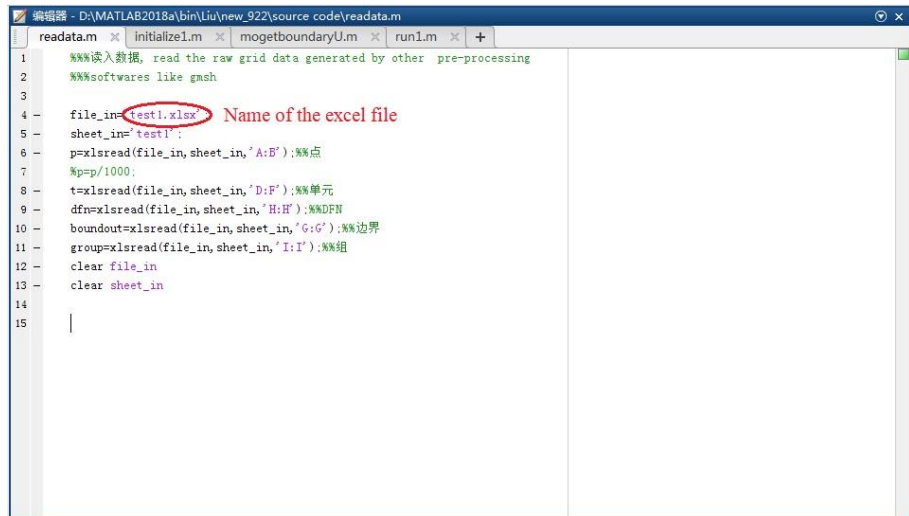


	A	B	C	D	E	F	G	H	I	J	K
1	0	0.025		343	174	171			0		
2	0.025	0		103	343	171			0		
3	0.05	0.025		232	94	355			0		
4	0.025	0.05		233	356	98			0		
5	0.025	0.025		147	79	256			0		
6	0.02	0.05		361	79	147			0		
7	0.03	0.05		93	151	308			0		
8	0.025	0.055		355	94	161			0		
9	0.03	0		356	163	98			0		
10	0.02	0		89	149	348			0		
11	0.025	-0.005		91	311	168			0		
12	0.000157	0.022201		141	168	311			0		
13	0.000627	0.019437		142	312	169			0		
14	0.001403	0.016743		143	313	170			0		
15	0.002476	0.014153		95	169	312			0		
16	0.003832	0.011699		97	170	313			0		
17	0.005454	0.009413		78	152	295			0		
18	0.007222	0.007222		92	201	151			0		

Fig.1 Data information in Excel

### 2.2 Data read using MATLAB

The script file named *readdata.m* in the source code folder is used for reading the excel data. It is worth mentioning that, the script does not need to be run separately because it is assembled in the file named *initialize1.m*.



```
1  %%%读入数据, read the raw grid data generated by other pre-processing
2  %%%softwares like gmsh
3
4  file_in='test1.xlsx' %%%Name of the excel file
5  sheet_in='test1';
6  p=xlsread(file_in,sheet_in,'A:B');%%点
7  %p=p/1000;
8  t=xlsread(file_in,sheet_in,'D:F');%%单元
9  dfn=xlsread(file_in,sheet_in,'H:H');%%DFN
10 boundout=xlsread(file_in,sheet_in,'G:G');%%边界
11 group=xlsread(file_in,sheet_in,'I:I');%%组
12 clear file_in
13 clear sheet_in
14
15
```

Fig.2 The *readdata.m* file in MATLAB

### 3 Running settings

#### 3.1 Running option

The running parameters that must be specified in the Y-Mat code include: the Number of Time Steps, the output frequency, and gravity. It is worth mentioning that, the time step size is calculated by the *timesize.m* file. And its formula is given by

$$\Delta t = \xi \min\left\{\frac{A}{C_p C}\right\} \quad (1)$$

Where  $\xi$  is safety coefficient range from 0.1-0.5;  $A$  is the element area;  $C$  is the element perimeter;  $C_p$ , the  $p$  wave velocity is given by

$$C_p = \sqrt{\frac{K + 4/3G}{\rho}} \quad (2)$$

Where  $K$  is bulk modulus,  $G$  is shear modulus;  $\rho$  is density of materials.

```

4  %<-----读取数据，reading data----->%
5  readdata;
6  %<-----定义全局变量并初始化----->%
7  global IE NewPoint IX Y rr mass Vol NewPoint0 damp plas U Acc geoxexist gacce Modeca sxx syy fst Ecenter
8  global CC ZZ Astore Interprop AAA opi sldis disdel dpsem
9  [NewPoint, IE]= moget1NewPoint1(p, t, group, dfn, boundout);
10 Modeca=1; %%1平面应力，2平面应变：1 represents plain stress problem, 2 represents plain strain problem
11 geoxexist=1; %%1 代表考虑地应力，0 代表不考虑地应力
12 sxx=0;
13 syy=0;
14 Circle=100000; %% total number of time step
15 count=0;
16 opi=5000; %% output frequency
17 gacce=0; %% gravity
18 disdel=zeros(IE,1);
19 dpsem=0.15;
20 M=IE*(IE-1)/2;
21 sldis=zeros(M,6); %%初始化位移矩阵，若单元数过大，可以分开生成，六个矩阵
22 al=0.63; bl=1.8; cl=6.0;
23 aal=al+bl;
24 U=zeros(3*IE,2); %%位移初始化，Initialization of displacement matrix
25 Vol=zeros(3*IE,2); %%速度初始化，Initialization of velocity matrix
26 Acc=zeros(3*IE,2); %%加速度初始化，Initialization of acceleration matrix
27 Boundxy=1; %%0是应力加载，1是位移加载：0 represents stress loading, 1 represents displacement loading

```

Fig.3 Running options

#### 3.2 Material assignment

The material parameters include: the element parameters and inter-element (initial crack or crack model) parameters, as shown in Table 1. And the parameters are assigned by two kinds of command in Y-Mat code.

$$name\_zset=groupzone(E,V,ro,C,fai,ft)$$

$$name\_inset=groupinter(C,fai,faires,ft,GI,GII,Pf,penn,pent)$$

Then  $name\_zset$  and  $name\_inset$  are renamed to the corresponding groups. As such, the parameters in the brackets are assigned to different groups.

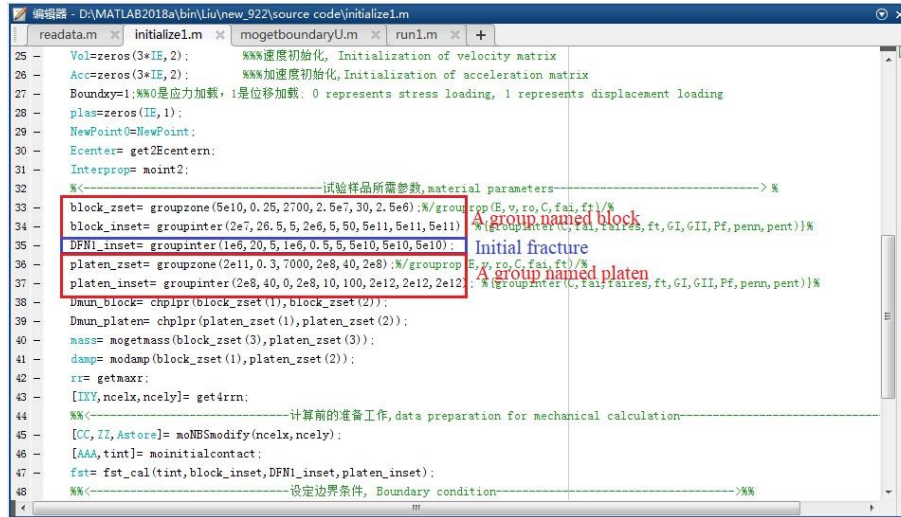


Fig.4 Material assignment

Table 1 Material properties and model parameters for mechanical test

		Rock sample	Loading platens
Elements	Young's modulus (GPa)	50	200
	Poisson's ratio (-)	0.25	0.3
	Density (kg/m <sup>3</sup> )	2700	7000
	Cohesion (MPa)	20	200
	Friction angle (°)	30	40
	Tensile strength (MPa)	5	200
Inter-elements (crack model)	Cohesion (MPa)	20	200
	Friction angle (°)	26.5	40
	Residual friction angle (°)	0	0
	Tensile strength (MPa)	2.5	200
	Mode I fracture energy release rate (N/m)	2	10
	Mode II fracture energy release rate	20	100
	Fracture penalty (Pa)	5e11	2e12
	Normal contact penalty (Pa)	5e11	2e12

### 3.3 Boundary conditions

Two types of boundary conditions including constant velocity and force can be applied to the boundary. If a boundary need to be fixed, a zero velocity are assigned. And the *mogetboundaryU.m* is the file that assigns boundary conditions. The specific format is:

$$F_{bound}=mogetboundaryU(a, Boundxy, xbou, ybou)$$

Where *a* can take 1 or 0. 1 means loading with platens; 0 means no loading platens, the velocity or force is applied directly to the boundary interface. *Boundxy* indicates stress loading (equal to 0) or velocity loading (equal to 1). *xbou*, and *ybou* are the magnitude of boundary conditions, respectively.

If the loading way is stress, the *mogetboundaryU.m* file should be calculated only once. And the result matrix *Fbound* should be added to total force in each step. And if the loading way is velocity, the boundary condition of corresponding nodes should be modified to the constant velocity each time step. So there is another file, *modifyVol1.m*, to complete this task, as shown in Fig.6.

```

1 %%boundary condition, stress or velocity
2 function Fbound=mogetboundaryU(a,Boudxy,xbou,ybou)
3     global NewPoint IE Vol Interprop
4     Fbound=zeros(3*IE,2); %%应力边界初始化
5     leftdownB=[min(NewPoint(:,1)),min(NewPoint(:,2))];%%左下边界
6     rightupB=[max(NewPoint(:,1)),max(NewPoint(:,2))];%%右上边界
7     diezou=1e-6; %%容差,tolerance
8     if Boudxy==0
9         sigmax=xbou; %%x方向边界力
10        sigmay=ybou; %%y方向边界力
11    else
12        volxB=xbou; %%x方向边界速率
13        volyB=ybou; %%y方向边界速率
14    end
15    if a==0 %%无加载盖, 可以加载应力和位移.no loading platen
16        if Boudxy==0 %%stress boundary condition
17            %<-----速度边界或应力边界赋值----->%
18            for ie=1:IE
19                for j=1:3
20                    if Interprop(3*(ie-1)+j,1)==1 %%边界界面
21                        h=j+1;
22                        if h>3
23                            h=1;
24                        end
25                    end
26                end
27            end
28        end
29    end

```

Fig.5 Boundary condition

```

1 %%modify velocity of loading elements in running
2 function modifyVol1(volyB)
3     global Boudxy Vol IE NewPoint
4     if Boudxy==1
5         for i=1:3*IE
6             if NewPoint(i,5)==1 %%上加载盖 upper loading platen
7                 Vol(i,2)=-volyB;
8                 %Vol(i,1)=0;
9             elseif NewPoint(i,5)==2 %%下加载盖 lower loading platen
10                Vol(i,2)=volyB;
11                %Vol(i,1)=0;
12            end
13        end
14    end
15 end

```

Fig.6 The correction function of boundary velocity

### 3.4 Mechanical calculation

All the above settings are assembled in the file, *initialize1.m*. Run the *initialize1.m* and *run1.m*, the procedure starts to run.

## 4 Post-processing in MATLAB

The Brazilian split test is a common and indirect way to measure the tensile strength of rocks or rock-like materials. As shown in Fig. 7, a circular disc with a 25 mm radius is loaded by two loading platens that move toward each other at a speed of 0.05m/s is established.

The results are saved as a file with a *.mat* extension, such as *50000.mat*. Each file contains new fracture, the displacement and stress information of the current time step.

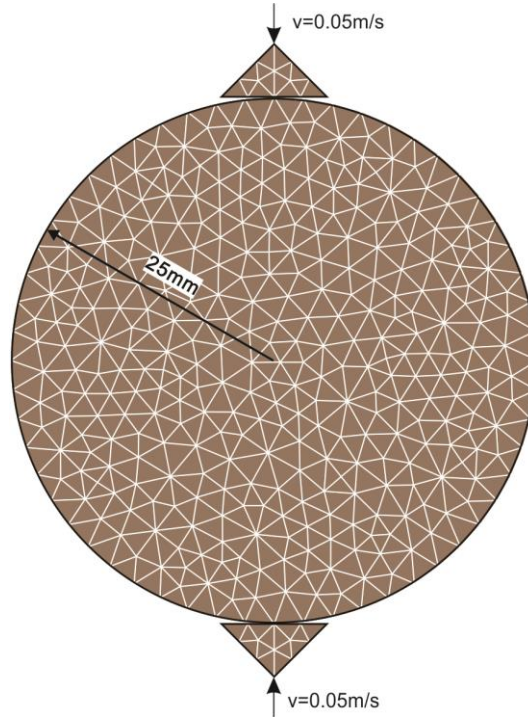


Fig. 7 Geometry, mesh topology and boundary conditions for Brazilian split test simulation

#### 4.1 Results of the failure trajectories and modes

The file *getimagen.m* can show the current position of the model with initial crack and new crack (if the cracks exist). As such, the failure trajectories and failure modes can be drawn in the figure. Inputting the command *getimagen(tint)* in the command window, the picture can be displayed, as shown in Fig. 8. What worth to mention is that, *tint* could not be changed, it's the internal data, not the parameter inputted by users.

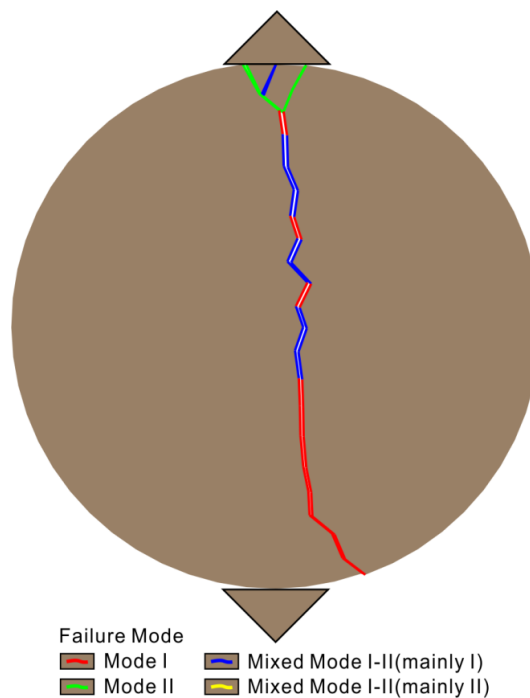


Fig. 8 Failure mode of the Brazilian disc

## 4.2 Results of the displacement of the model

The file *plot\_dis.m* can show the displacement cloud of the model. Inputting the command *plot\_dis(xys\_index)* in the command window, the picture can be displayed, as shown in Fig. 9. The *xys\_index* must be change to 1, 2 or 3. For example, *plot\_dis(1)* is an appropriate command. 1 represents the X-direction displacement component; 2 represents the Y-direction displacement component, as well as 3 represents the total displacement.

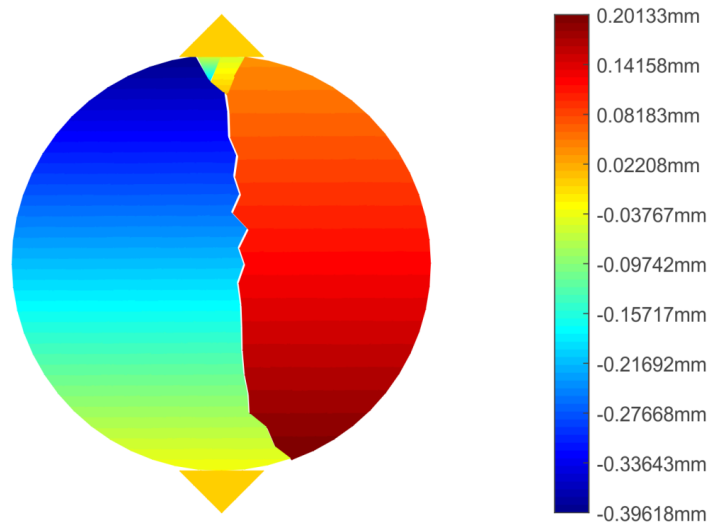


Fig. 9 Contour plot of x-component of nodal displacement in the model

## 4.3 Results of the stress distribution of the model

The file *plot\_sig.m* can show the stress cloud of the model. Input the command *plot\_sig(xys\_index,sigmaxy)* in the command window, the picture can be displayed, as shown in Fig. 10. The *xys\_index* must be change to 1 ,2 ,3, 4 or 5. For example, *plot\_sig(2,sigmaxy)* is a right command. 1 represents the X-direction stress component; 2 represents the Y-direction stress component; 3 represents the shear stress; 4 represents the min principal stress, as well as 5 represents the max principal stress.



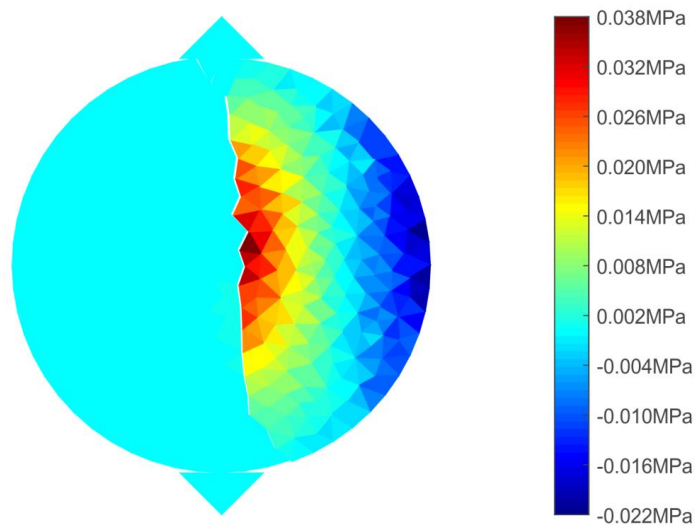


Fig. 10 Contour plot of vertical stress in the model

MATLAB has the advantages in numerical calculations and graphics processing. Meanwhile, errors can be found easily due to the readability of the code and data. A GUI of Y-Mat should be made to conveniently read the grid data and input the parameters of the model in the future work.