

Y-Mat Code

A brief User Guide

November 2018

Gang Liu

Key Laboratory for Geo-hazards in Loess Area, MNR
Xi'an Center of China Geological Survey/ Northwest China Center for Geoscience Innovation
No. 438, Youyi Eastern Road, 710054, Xi'an, Shaanxi, P.R.China
E-mail address: liugang_iggcas@163.com (G. Liu)

Table of Contents

1 Introduction	1
2 Pre-processing settings	2
2.1 Data preparation	2
2.2 Data reading using MATLAB	2
3 Running settings	3
3.1 Running options	3
3.2 Material assignment	4
3.3 Boundary conditions	5
3.4 Mechanical calculation	6
4 Post-processing in MATLAB	6
4.1 Results of the failure trajectories and modes	7
4.2 Results of the displacement of the model	7
4.3 Results of the stress distribution of the model	8
4.4 Results of the stress contour 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
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 ν into bulk modulus K and shear modulus G
calculate.m	Calculate the contact force of the discrete elements
getmaxr.m	Calculate the data for NBS method
get4rrn.m	Calculate data for NBS method
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
getFdn.m	Calculate the deformation forces matrix of elements
chplpr.m	Calculate the elastic matrix according to the elastic modulus E and Poisson's ratio ν
geostress.m	Apply the geostress to the model
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
extran.m	Code for excavation, set the excavation area by group number
exchange.m	Code for excavation, to free the interaction of joint elements in excavation area
run1.m	The main program
stepsize.m	Calculate the time step size using Young modulus E , poisson ratio ν and density ρ
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)

stresscont.m	Plot the stress contour
--------------	-------------------------

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 reading 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*.

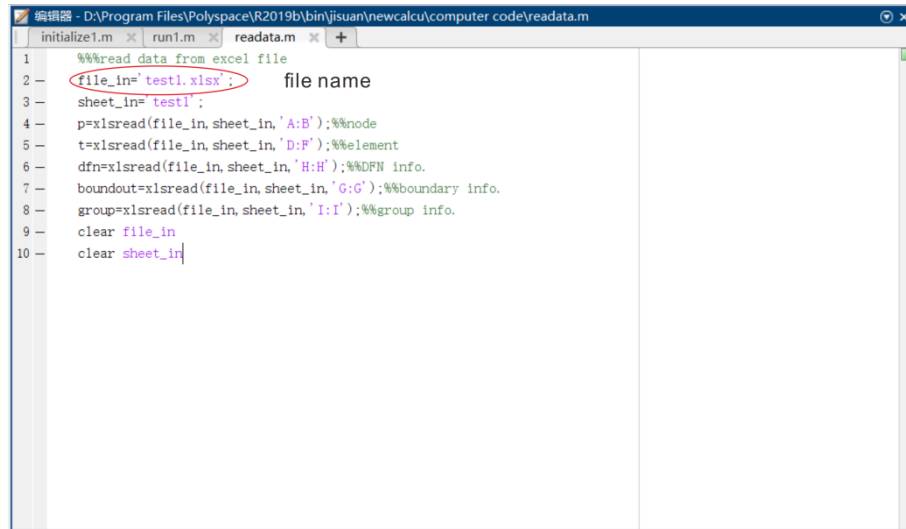


Fig.2 The *readdata.m* file in MATLAB

3 Running settings

3.1 Running options

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 *stepsize.m* file. And its formula is given by

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

Where ξ 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; ρ is density of materials.

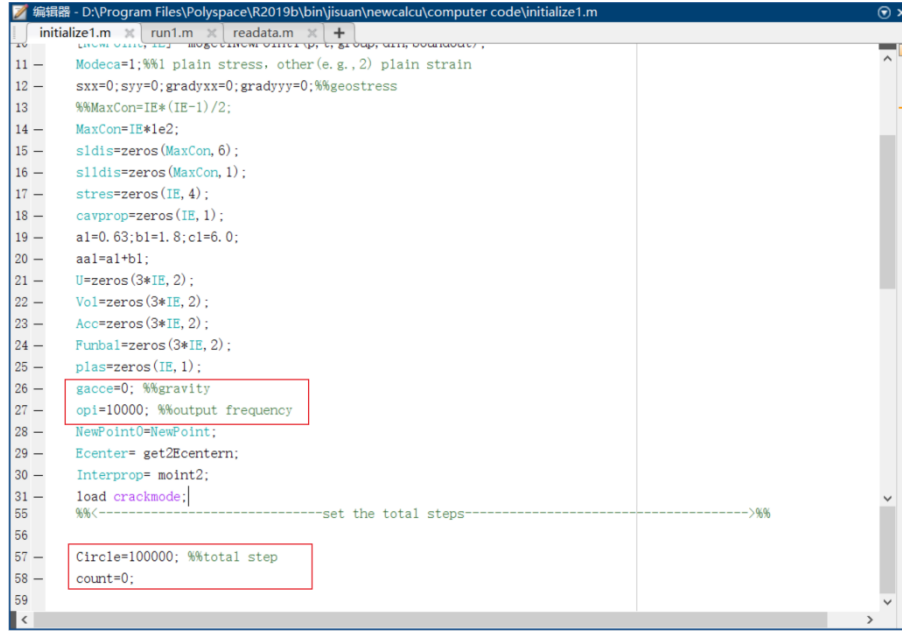


Fig.3 Running options

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.

$$name_zset=groupzone(E,V,ro,C,fai,ft)$$

$$name_inset=groupinter(C,fai,faires,ft,GI,GII,Pf,penn,max_friction)$$

Then *name_zset* and *name_inset* are renamed to the corresponding groups. The meanings in the first brackets are young modulus, poisson's ratio, density, cohesion, friction, and tensile strength of the finite elements. The meanings in the second brackets are cohesion, sliding friction, Residual friction, tensile strength, mode I fracture energy release rate, mode II fracture energy release rate, fracture penalty, normal contact penalty, and maximum static friction of the joint elements. As such, the parameters in the brackets are assigned to different groups.

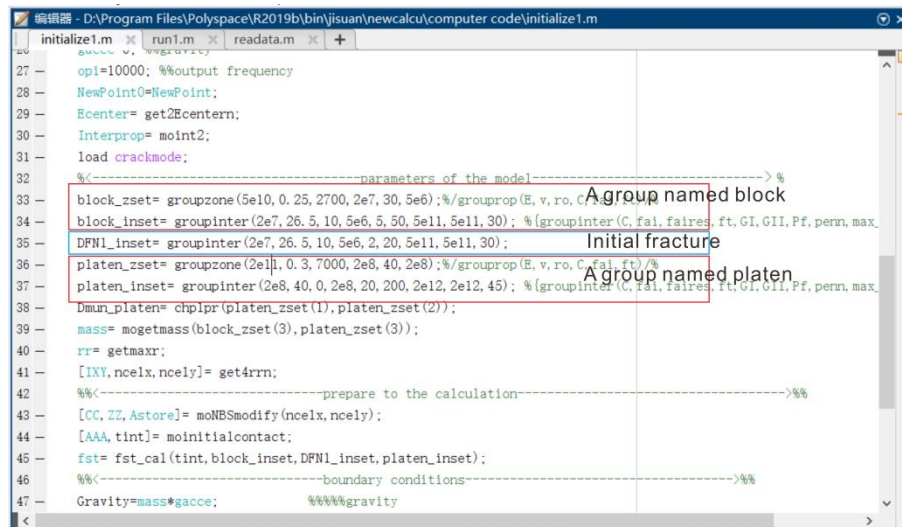


Fig.4 Material assignment

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 ³)	2700	7000
	Cohesion (MPa)	20	200
	Friction angle (°)	30	40
	Tensile strength (MPa)	5	200
Inter-elements	Cohesion (MPa)	20	200
	Maximum static friction angle (°)	30	45
	Sliding friction angle (°)	26.5	40
	Residual friction angle (°)	10	0
	Tensile strength (MPa)	5	200
	Mode I fracture energy release rate (N/m)	5	20
	Mode II fracture energy release rate	50	200
	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:

$$F_{\text{bound}} = \text{mogetboundaryU}(a, \text{Boundxy}, x_{\text{bou}}, y_{\text{bou}})$$

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). x_{bou} , and y_{bou} 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 F_{bound} 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.

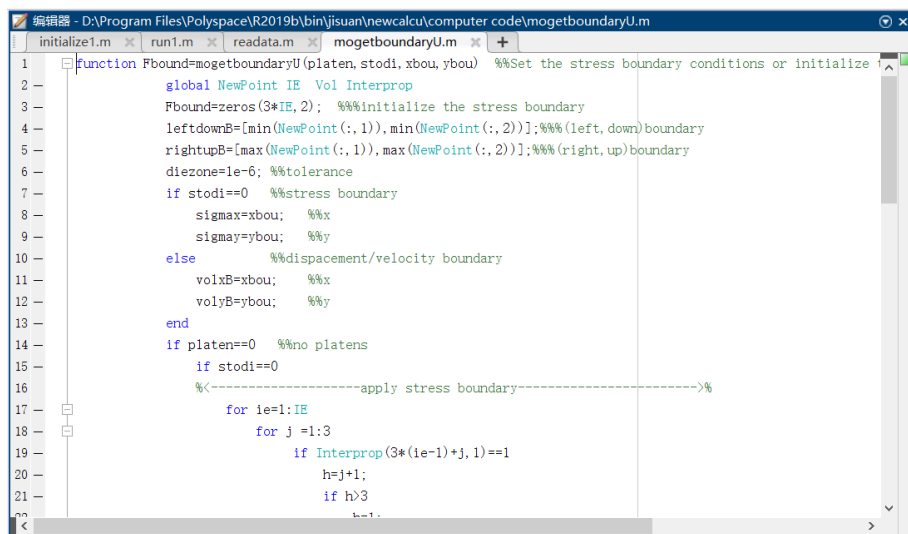


Fig.5 Boundary condition

```

1 function modifyVol1(volyB) %%apply the velocities to the boundary
2     global Vol IE NewPoint
3     %if Boundxy==1
4     for i=1:3*IE
5         if NewPoint(i,5)==1
6             Vol(i,2)=-volyB;
7             Vol(i,1)=0;
8         elseif NewPoint(i,5)==2
9             Vol(i,2)=volyB;
10            Vol(i,1)=0;
11        end
12    end
13    %end
14 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 *100000.mat*. Each file contains new fracture, the displacement and stress information of the current time step.

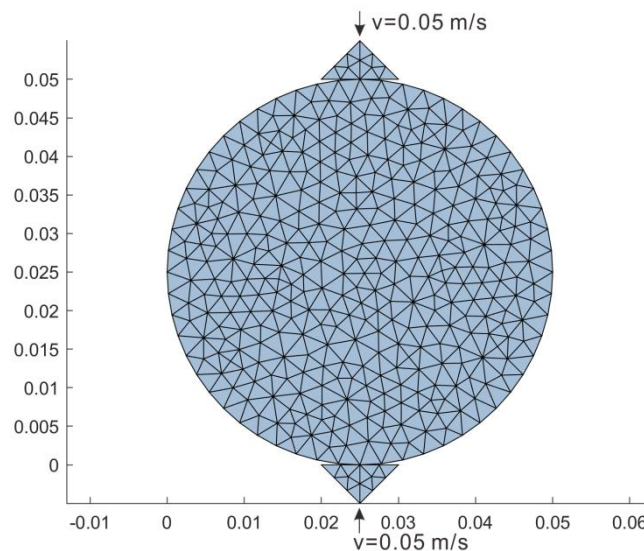


Fig. 7 Geometry, mesh topology and boundary conditions for Brazilian split test simulation

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,gridshow)* in the command window, the picture can be displayed, as show in Fig. 8 (*getimagen(tint,0)*). *Tint* is the internal data, so no need to enter manually; *gridshow* represents whether to display the grid, 1 is displayed, 0 is not displayed.

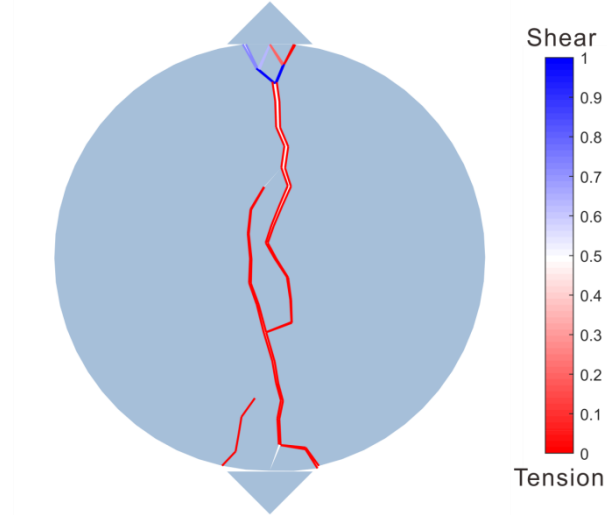


Fig. 8 Failure mode of the Brazilian disc

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.

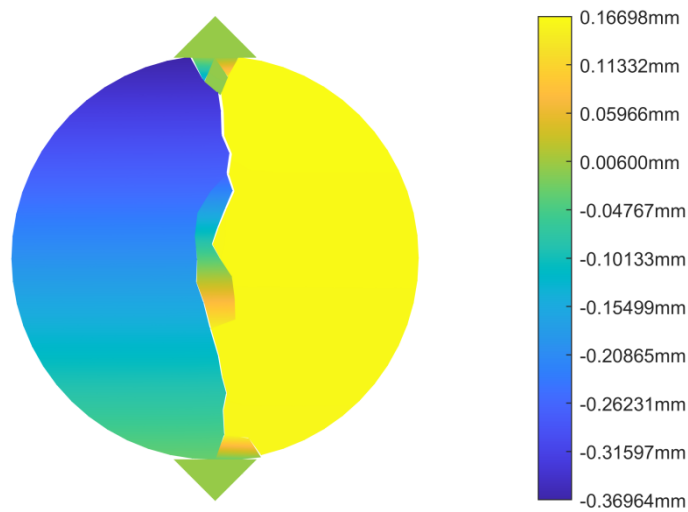


Fig. 9 Cloud plot of x-component of nodal displacement in the model

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)* 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)* 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.

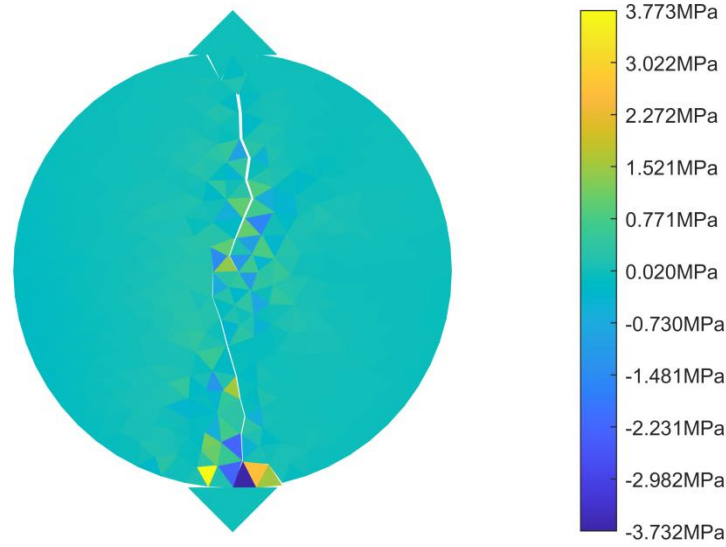


Fig. 10 Clond plot of vertical stress in the model

4.4 Results of the stress contour of the model

The file *stresscont.m* can show the stress cloud of the model. Input the command *stresscont(xyt_index)* in the command window, the picture can be displayed, as shown in Fig. 11. The *xyt_index* must be change to 1, 2 or 3. For example, *stresscont(1)* is a right command. 1 represents the X-direction stress component; 2 represents the Y-direction stress component; 3 represents the shear stress.

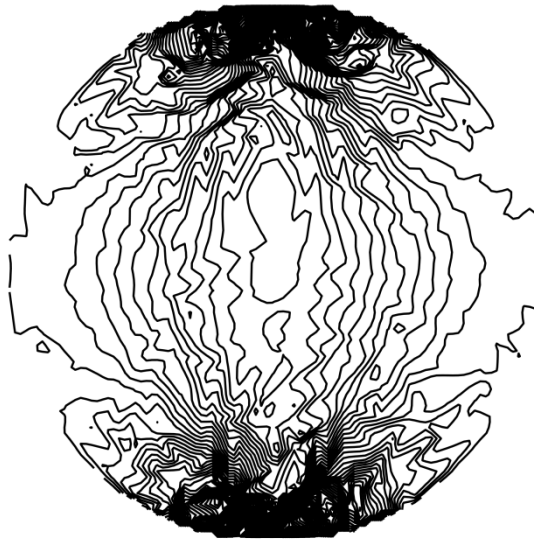


Fig. 11 Contour plot of x-component stress in the model

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.