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 (G. Liu)

Table of Contents

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 possion ratio v 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
mofracture.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 v
mofrd.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, possion ratio v 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)

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.

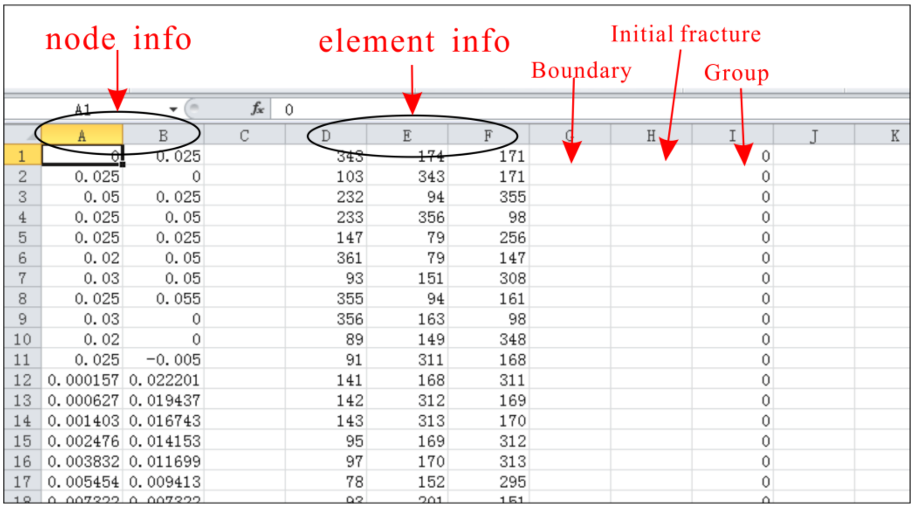

Fig.1 Data information in Excel

2.2 Data read using MATLAB

The script file named *readata.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*.

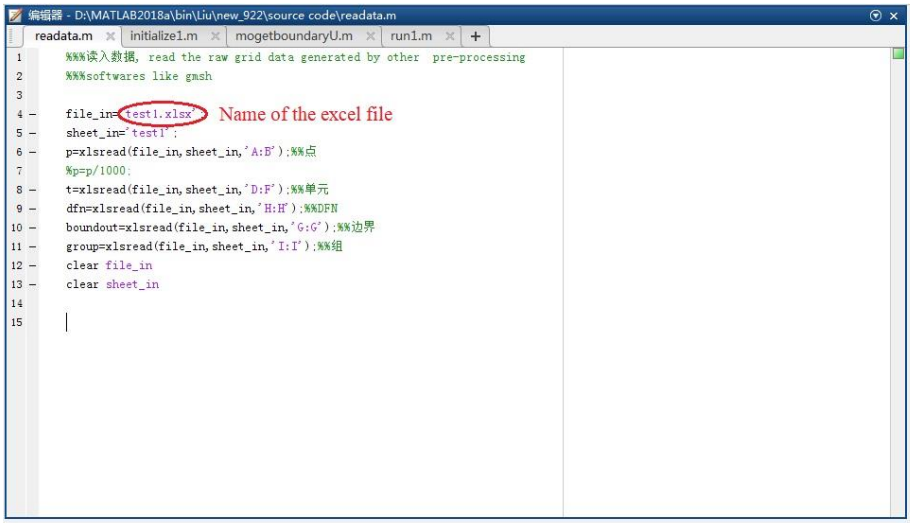

Fig.2 The *readata.m* file in MATLAB

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

$$\Delta t = \xi \min\{\frac{A}{C_p C}\} \tag{1}$$

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

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

Where K is bulk modulus, G is shear modulus; ρ is density of materials.

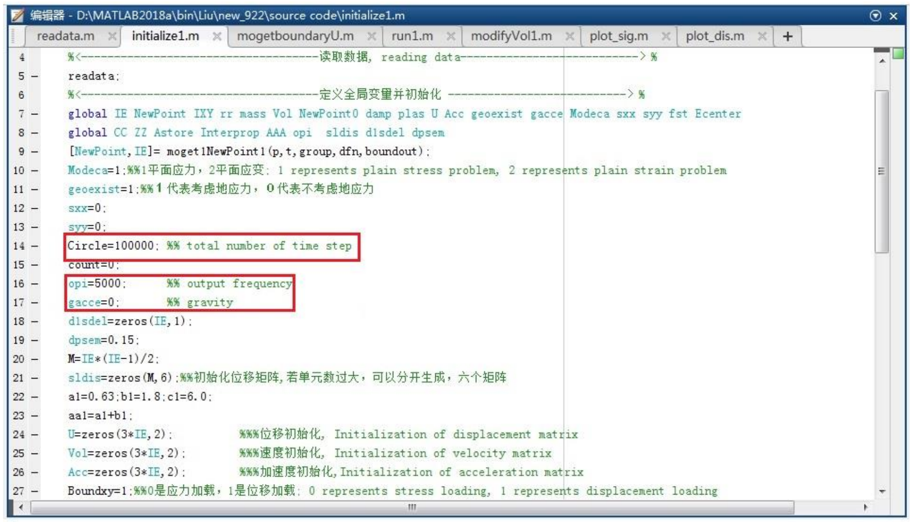

Fig.3 Running options

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,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.

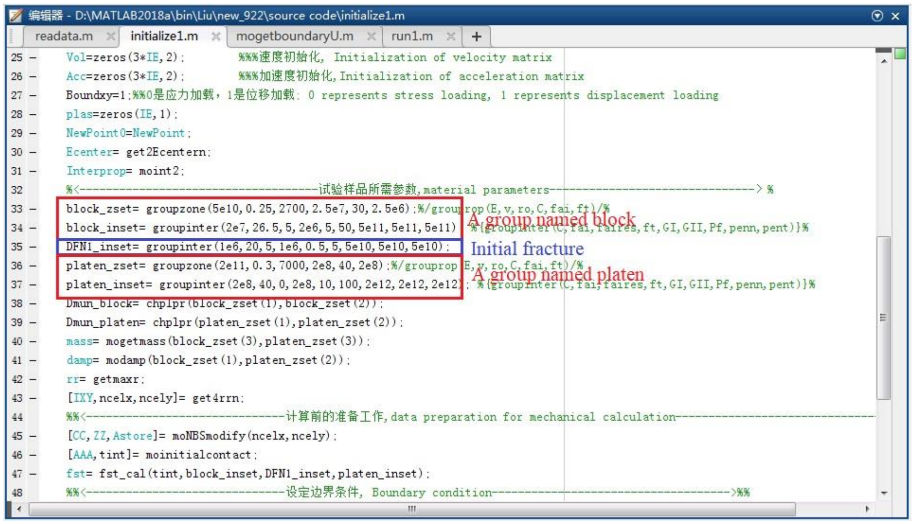

Fig.4 Material assignment

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^3)	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:

$$Fbound=mogetboundaryU(a, Boundxy, xbou, ybou)$$

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.

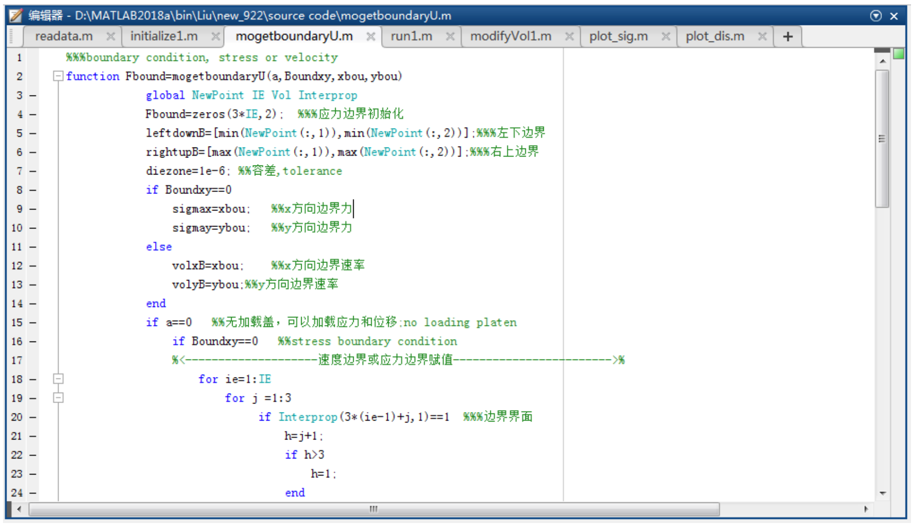

Fig.5 Boundary condition

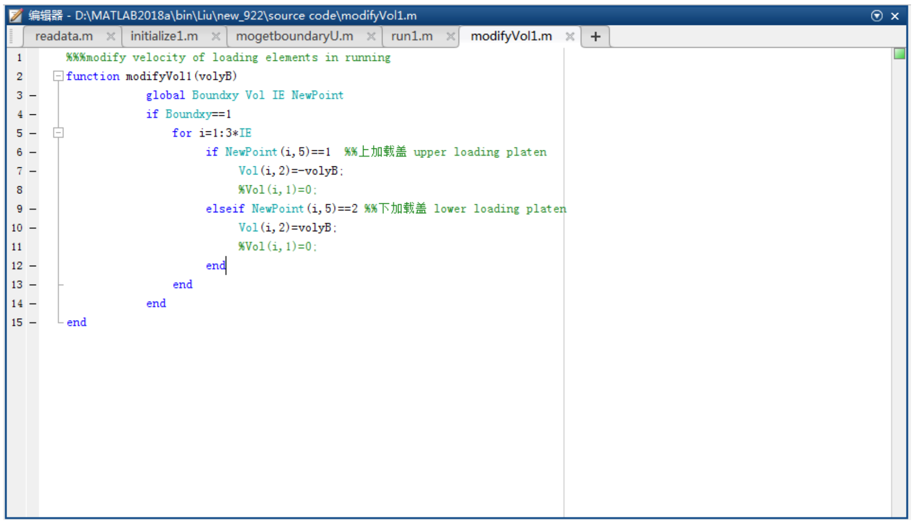

Fig.6 The correction function of boundary velocity

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.

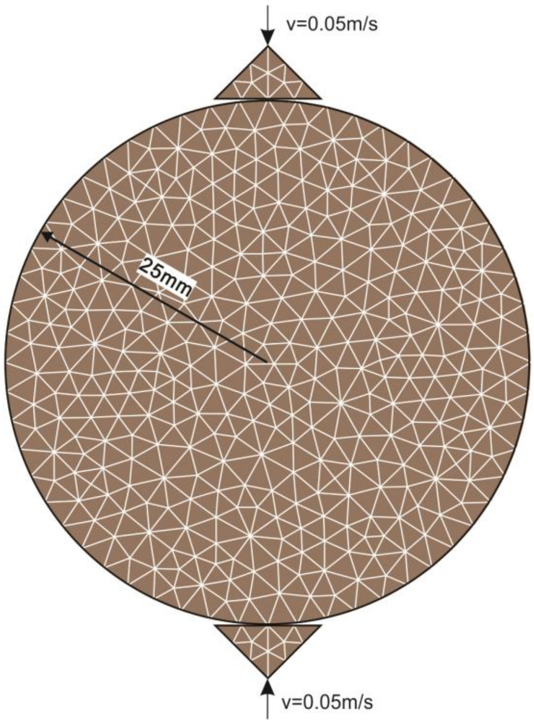

Fig. 7 Geometry, mesh topology and boundary conditions for Brazilian split test simulation

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 show 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.

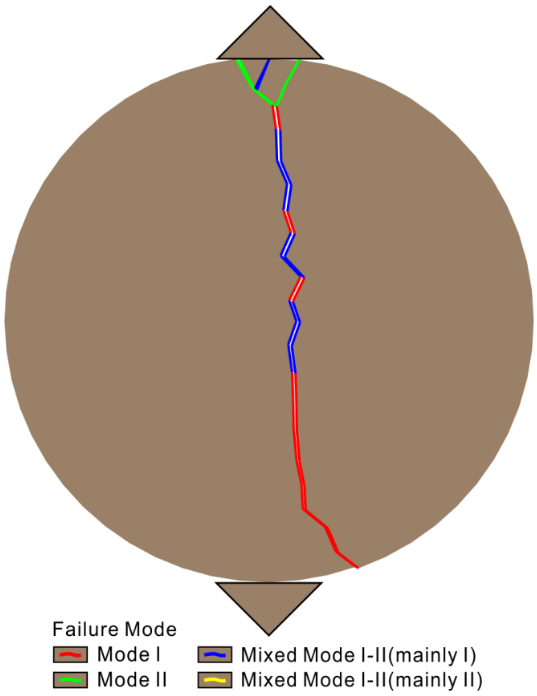

Fig. 8 Failure mode of the Brazilian disc

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.

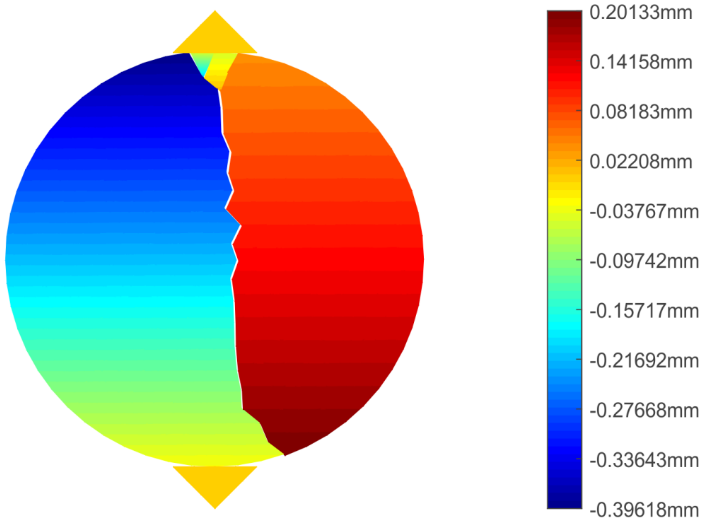

Fig. 9 Contour plot of x-component of nodal displacement in the model

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.

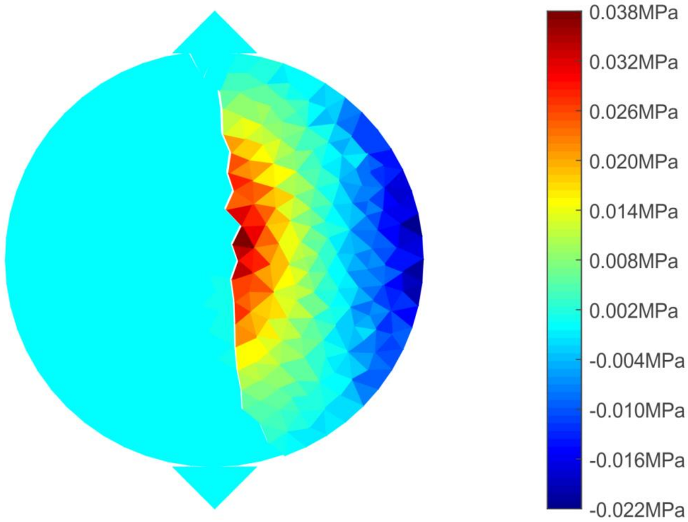

Fig. 10 Contour plot of vertical stress in the model

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.