

GRAVITY MODELLING AND INTERPRETATION

Objective - Use the interactive modelling package "MAGIX-plus" to become familiar with the characteristics of gravity anomalies for some simple models. Then, use MAGIX-plus to interpret a gravity anomaly over a sulphide ore deposit using a "cut-and-try" iterative modelling process.

1. Gravity Modelling Study

Familiarize yourself with 2 and 2.5D modelling concepts and the modelling program MAGIX-plus by investigating the gravity response of a dyke-like body (thickness $\ll$ depth extent) of long strike when changing depth of burial and dip (Lab Notes Fig. 1.7, pg. 68). Then, for one of the modelled cases, investigate the effect of changing a) the strike length and b) the thickness. Plot a few representative models for each study. Describe how variation of the model parameters affects the response. How sensitive is the response to change in strike length? In regard to a body's gravity anomaly, what is a short strike length? a large strike length? an infinite strike length? What effect does changing the thickness and/or the density contrast of the body have?

Start with the file P2G.MGX. Choose realistic densities and sizes, and remember that 0.1 and 1.0 milligals (1 and 10 $\mu\text{N/kg}$) respectively are about the smallest anomalies that can be recognized and measured in local (~ 50 m station) and regional (~ 1 km station) spacing surveys. Note that the visual cross section of the model may not have 1:1 vertical : horizontal scale.

2. Gravity Interpretation Study

In file MOBRUN.MGX, you are given the principal profile of the residual gravity over a sulphide ore deposit in the Abitibi region of Quebec. This area is covered with a few tens of meters of glacial deposits (typical density about 1.7-2.2 Mg/cubic m). They lie on deformed Archean metavolcanic and metasedimentary rocks that typically have densities in the 2.7 – 2.9 Mg/cubic m. An electromagnetic survey has discovered a highly conductive feature in the bedrock that is possibly due to a concentration of sulphide ore minerals in the metavolcanic rocks (iron and base metal sulphide minerals such as pyrite, pyrrhotite, chalcopyrite, galena, etc. Sulphides are high density minerals, and, if present at high concentration can raise the rock density to 3.5 – 4.5 Mg/cubic m (for a concentration of around 70 vol %). The initial drill hole did encounter sulphide mineralization. A detailed gravity survey was therefore carried out to estimate the extent of the deposit.

You are provided with a map of the *residual* Bouguer gravity anomaly over the area. This is the Bouguer anomaly after an estimate of the background response from the normal geology has been removed. You can estimate the strike length of the anomalously dense formation from the map and then interpret the feature's cross-sectional form by trying to match the anomaly's principal profile with the response from a polygonal model.

The potential field modeling package "MAGIX-plus"

1. Magix-plus permits interactive forward 2D (section) modelling and also inversion of both gravity and magnetic data. A 2D model is described by a polygonal cross-section and a strike length – either infinite (pure 2D) or finite (2.5D or "2.75D"). Strike length is defined by two distances, one into and one out of the screen. If the strike lengths in and out are equal, the model is described as 2.5 D, if they are unequal, the model is "2.75 D" in the MAGIX jargon. The package also allows you to include uniform layers, i.e., horizontal slabs, which extend to infinity in all directions. These affect only the base level of gravity profiles, and they have no effect in magnetics. You must assign a density (susceptibility and magnetization in magnetics) to each body and to the host medium. The program uses only the density differences, so you may define the host medium to have zero density if you wish to label other bodies in density contrast. In modelling magnetics, you must supply additional information
2. There are a lot of features available to the interpreter in the Magix package, but only a few are required for the lab exercises. These notes give you the minimum commands required. If you wish to explore other features, ask the demonstrator for the MAGIX Users' Manual. Please note that the current version of the program is an updated version of the one described in the Users' Manual.
3. Login to the computer using the student/student account/password. To start Magix, double click the Magix icon in GpLab icon or go to the start area on the lower left of the screen. You are also dropped into D:\USERS\TEMP, i.e. all files you read from or write to are in that directory unless you change the default directory manually (under the EXIT heading). WHEN YOU FINISH, PLEASE CLEAR THIS FOLDER TO THE TRASH.

The Magix package operates in one of three modes:- menu, command, or interactive (interactive uses the mouse and keypad to move things around and change model parameters). You will only need to use the menu and interactive modes. The menu mode moves you about within the package, e.g., going from input procedure to data editing procedures to exiting the program. Most of the modelling and interpretation is done in the interactive mode. Because there are many options, the program seems very cumbersome at first. Get used to exploring it by accepting defaults by keying <return>.

4. The first screen to pop up is a license notice ... just hit return. The next screen asks for the default drive ... just hit return. You are now presented with a matrix system of system commands/menus - you are in the menu mode. Along the top of the screen are the "categories", each of which has an associated menu which pops up when the category is highlighted. You move from category to category using the (cursor move) left and right arrow keys or the mouse. Within each menu you move through the list using the up and down arrow keys or mouse. When a menu selection is highlighted, press return to enter that mode of operation. The six categories that will be required for this exercise are:

EDIT INPUT OUTPUT PLOT CALC EXIT

They are described below in the order you will likely encounter them:

INPUT

You are expected to have some sort of field data set that you want to match by modelling and to have copied it to your working directory (D:\USERS\TEMP). Even if you don't, you will need a dummy set to tell the program where the observation points should be. The easiest way to create a new input file is to edit an old one. So move the cursor over to INPUT and select (highlight) the get (data+model) menu option ... press <return>. You are then prompted for the data set file name ... enter one of the following files (you should already have put them in directory D:\USERS\TEMP):

For example: P2G.MGX for the type curve modelling exercise (1a).
 MOBRUN.MGX ... for the gravity interpretation exercise (1b).

The filename extension ".MGX" is the default value so it is not necessary to include it. The ".MGX" files are binary files which contain all of the information required for modelling and interpretation; i.e. the survey parameters, the model parameters, and the observed (field) data. In essence, they are a snapshot of the state of the modelling/interpretation session at the time the file was saved. You will not need to use any of the other INPUT menu options. Note that the name of the current .MGX file is shown at lower left corner of the screen. If there is a problem finding the files, check the "path" set for the program under EXIT/ SET new path.

CALCULATE

The package works best in interactive mode. Move the cursor over to the CALCULATE category and select the interpret (interactive graphics) menu option ... press <return>. The screen now displays the model cross-section (if one has already been created) as a purple polygon and graphs the field data as blue squares. The green horizontal line represents the ground surface. Note the vertical scales ... magnetic data are presented in nanoTeslas (gammas), gravity data in milliGals (one milliGal = 10 micrometers / s² ... the depth is in kilometres. The horizontal scale may be different from the vertical scale and is also expressed in kilometres. The calculated response curve is not yet shown.

Two lines at the bottom of the screen show the status of the interactive session (current data set, vertical exaggeration in the plot, x z position of cursor in screen coordinates, current body under the cursor and its attributes) and a table of single character commands that invoke various model (polygon) and computational parameters. Once again there is a hierarchy of options, so that each time a single character option is selected (press the key without a <return>) a new table of options is displayed. The options are more or less self-explanatory. It is suggested that you move the mouse with your right hand and enter the single character commands from the keyboard with your left hand. You will be mostly using the following commands:

Outermost table

^ -> insert body	to add a new body
P -> replot	to refresh screen or reset the plotting scales, which will be necessary if the calculated response won't fit on the screen.
E -> edit bodies	to change the polygon, e.g. rotate, translate, move vertices, etc.
B -> edit body params ..	to change the density, strike length, etc.
F -> forward calc	to calculate the response of current model. After calc has been done once, the response is updated in real time.
	If response gets too large, it may be necessary to do P -> replot.
Q -> quit	to exit to the main "categories" menu

(Details of E and V continue on next page)

within E (the edit bodies table)

- -> delete body to delete a body (polygon) "captured" by the crosshairs*.
- V -> edit vertices to add, delete, move vertices.
- P -> replot screen refresh.
- Q -> quit to exit to the outermost table.

within V -> (the edit vertices table)

- -> delete vertex to delete a vertex captured by crosshairs .
- + -> toggle insert vertex ... to "spawn" a new vertex from a "captured" vertex*,
toggles each time "+" is pressed.
- P -> replot screen refresh.
- Q -> quit to exit to the E -> edit bodies table

To capture a polygon or a vertex move the crosshairs over the target and click any mouse button once. To release the target, click the mouse button again.

Magix draws a graph of the observed and modelled data (and differences if requested) and a cross-section of the model. It generally uses autoscaling for this, which can be confusing and annoying at times, especially on the model section. The scaling can be controlled, but the menus for this are only accessible in the command mode. See the manual for details.

OUTPUT

Once you are happy with the model, save the current state of the modelling session by quitting (Q -> quit) the interactive mode so that you are once again in the main "categories" menu. Then select the "rename" (new data set name) item in the OUTPUT category menu. Enter a unique name at the prompt. We suggest using your abbreviated name plus a number (or name and number) that identifies the trial. Remember this will be snapshot of the current state of your modelling session. Each time you give a file name, it is named in the lower left of the screen changes to your new file name. But note that the file has not yet been saved to disk! Select save (new binary file) to do this. Save differs from replace in that Save only writes to disk after the data set filename has been renamed. It is not possible to overwrite an existing data set using save. Replace, on the other hand, will overwrite the current data set and should only be used if you really don't like the previous model.

PLOT

To get a hard copy of the current modelling session (i.e. what you see in the interactive modelling screen), select the PLOT (hard copy graphics BOTH) item in the PLOT category menu. If you are prompted for a file name, just enter PRN.DMP. The plot will then be output from the HP Laserjet printer. Please make sure that the plot is complete. MAGIX has a habit of leaving off the labels for the vertical axes when the numbers (calculated response) get too large and they run off the edge of the normal plotting area. When this happens, the laser printer will hang up with a print overrun error ... press reset to put it back in ready state. You will have to try using a smaller density or susceptibility in your model so that the response is not over 9999 milliGals or nanoTeslas. (These are extreme values in gravity, but not unheard of in magnetics).

EDIT

You can edit the data set manually. As mentioned above, the data set contains all of the information required to reinstate your modelling session. The data set consists of five data groups. The EDIT menu gives you the option of editing these five groups individually or the two most important ones together (stepping through each one by one) using a spreadsheet-type editor. The principal EDIT functions (menu items) and their data sets are:

EDIT DATA edit these (*) groups together;
 EDIT DATA PARAMETERS* edit the general survey parameters;
 e.g. type of survey (gravity or magnetic), survey location, units used to describe the grid
 (kms, feet, miles), orientation of the profile and body strike direction(s) w.r.t. true north,
 local magnetic field inclination, declination and intensity
 EDIT DATA VALUES* edit parameters for all polygons;
 e.g. polygon vertices in x z coord pairs, strike length in and out, density/susceptibility,
 remanent magnetization
 To edit an individual polygon use EDIT BODY [BODY NUMBER].
 EDIT REGIONAL Edit the regional field profile, which has been removed from
 the observed data (manually entered);
 EDIT MODEL LABELS edit polygon labeling information.

This editor mode is really only necessary when you wish to change the regional (background) response and the mode of calculation (magnetic or gravity). All other parameters are easily changed in the interactive mode.

EXIT

To exit from the package, move the cursor over to EXIT and select (highlight) the exit (return to system) menu option. When prompted, answer yes or no. To temporarily exit to an emulation of the DOS command environment, select the shell (DOS command shell) menu option. You can then execute all normal DOS commands, e.g. DIR, DEL, REN, etc. To return to MAGIX, enter an EXIT command and confirm it with Y. In some cases, you may have to repeat the procedure. The first request to exit seems sometimes to be rejected (bug).