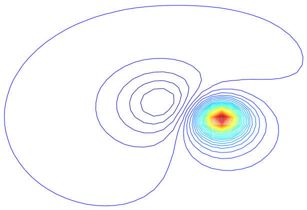


LAB 7

GRAVITY MODELING WITH GM-SYS

GM-SYS is a 2 3/4 D gravity and magnetics modeling package. GM-SYS can model Free-Air or Bouguer anomalies. If you model Free Air anomalies, GMSYS will do a 2 D terrain correction for you. If you model Bouguer anomalies, it is easy to see the gravity anomaly you are trying to model. With Bouguer anomalies, it is best to use the terrain density in the Bouguer correction and not the defined value. You must also make the air block the same density as the topography.

Instructions for creating a GMSYS project are on the last page of this handout. Once you have created a project, you just need to start MONTAJ, load GMSYS into MONATJ and open the project for work. The blank GM-SYS screen comes up, along with an *ACTION* menu bar at the top.

The top panel has a plan view of the model, at some depth slice. Once a model gets beyond a few blocks, you will find the plan view very useful in keeping track of what the model actually looks like. Also, if you are working in 2 3/4 D the plan view lets you see if you have set the azimuth, strike and off-profile distances properly. The bottom panel is a section through your model beneath the profile. The second panel from the top contains the observed mag data, and the predicted mag from the model. You can reduce the thickness of this panel to zero, if you have no mag data to look at. The third panel from the top has the observed and predicted gravity data.

The green dots in the mag and gravity panels are the observed data, and the thin line passing almost through the green dots are the gravity predictions calculated from the model. Also shown is a blue line, which is the difference between the observed and predicted data. This is also called the DATA MISS-FIT. The standard deviation of the data misfit is also shown in the bottom left of each data pane. You can easily construct an arbitrarily complex model which will result in predictions that exactly match the data, ie the standard deviation of the misfits is zero. However, a very important principle of modeling is that you want to find the **simplest** model that fits the observed data as well as the observed data has been observed. That is, the standard deviation on the data misfit should be about the same as the standard deviation on the repeats. Of course, you should also try to get data misfits that look random. Any correlation of data misfit with observed gravity, or in fact, any trend at all in the data misfit needs to be addressed by changing the model.

In the model pane, the green triangles represent the location of stations, and the red dots (joined by straight lines) are vertices of the model. Since GM-SYS will be doing a terrain correction the model at the surface needs to be defined to a few cm. Try moving one of the vertices that define the model, by clicking on the move point symbol in the *ACTION* menu, then clicking on a vertex, holding down the mouse button and dragging the point. You should see the predicted data, and the misfit line, change as you do this. Release the mouse button and the point is fixed at a new location. The *ACTION* menu has buttons for adding points to the model, or deleting them. Try this. The EYE icon in the action menu is the *EXAMINE* button. Click on this, then on some area in the model. The block of material that has the same density, or magnetic properties as the point clicked on will be outlined, and a menu will pop-up into which you can type changes in density etc. Each block can also be filled in with a pattern or color from this menu. The *ACTION* menu also has buttons for zoom-in, zoom-out, and box zoom. To use zoom-in and out click on the button, and then on a point in the data or model panels. To use box zoom click on the button, and then define a box in the data or model panels by clicking at the position of the lower left corner of the desired box and dragging to the position of the upper right corner.

The *ACTION* menu itself can be canceled, and retrieved again by clicking on the DISPLAY pull down menu at the top of the screen and activating or de-activating the *ACTION* toolbox. Any of the functions on the *ACTION* menu can also be accessed through the menu at the top of the screen.

Clicking with the right mouse button in any of the panels pops up a menu of commands to be executed in that pane. It would be a good idea to go through this now, before attempting to read in and work with your own data. Exit the demo by clicking on the x in the top right corner (not the x to exit GM-SYS).

To read your data into GM-SYS you will need to create an ASCII data file. In this file you need three columns, one for positions along profile, one for elevations, and one for Free-Air or Bouguer gravity anomalies. You can model either free air or Bouguer. If you model free air anomalies, most of the gravity variance you see may well be the attraction of the topography. If you model Bouguer ypu have to be aware that since you have already done an attraction for topography and the data still have elevations you need to have the air block the same density as the terrain so that there is no density contrast at the surface. Otherwise, GM-SYS will be doing a second terrian correction. You may also have several lines at the top describing the data. The files fs????.txt are either free air or Bouguer gravity data collected over the years in the field school area.

In the menu at the top of the screen click on the blank page icon and then click on *new model.* A dialog box will pop up into which you can enter information, such as min and max profile x, min and max depth etc. The profile azimuth is the direction of the positive direction of the profile measured in deg from N. Thus an EW profile (+E) would have an azimuth or 90 deg. The model relative strike is measured in degrees clockwise from the positive direction of the profile. In the middle of the *create new model* menu are sections to import topography and gravity data. Import both the gravity and the topography, even if the topography data is in the same file. This will ensure that the top of the default model conforms to the topography. After you click *import from file* Click on *file* and go through the menu until you see the data file you have prepared. Highlight this file in the list and click open. The file will be displayed - make sure it is OK - and hit OK. Enter the various parameters the menu calls for - number of lines at the top of the file, the units of measurement, the sign convention, (+ up).

Once you have the data read in click *CREATE* at the bottom of the *Create new model* menu. This will create the GM-SYS workspace that you used before, only now you have the default model, which is a single block of crustal material as thick as whatever you specified in the new model creation and extending to ± 30,000 km in the profile direction, embedded in air. If you read in a topography file, you should see that the default block conforms to this surface. Hit *zoom out* several times until you can see all of the green dots in both the model and data panels, and then box zoom in on both to see if everything looks OK. Are the topography and gravity right side up?

Click on *view* in the top menu, and then on *infinity* to zoom all the way out. Your default model extends $\pm 30000 km$ either side and is 30 km thick. It is a featureless slab, except for the profile, where it should conform to your surveyed elevations. Click on **EXAMINE,** (the **EYE** icon) and then on the model and make the density whatever you found from your terrain analysis. Exit EXAMINE, and click **COMPUTE** in the top menu. This recomputes the gravity anomaly from your model. Click on **PROFILE** the **GRAVITY** to view a table of observed and calculated gravity. **SAVE** the model. GMSYS will write a bunch a files to disc with whatever name you chose. The one with extension .grv contains the table you just viewed. Edit this table to a column format that Matlab can read and load it into Matlab. The calculated gravity has a 2D, or 2 3/4 D terrain correction built in. To see how important this is do a simple Bouguer correction on the calculated gravity with the same density you used in GMSYS. The residuals are the terrain corrections used in GMSYS.

Now you should see the whole default block. Click on *add points* and add a point on either side of the default block just below the top. Now click on *split block* and then double click one of the new points and drag it over to the other one. This should create a new line, which splits the default block in two. The lower one should be called COPY OF CRUSTAL BLOCK. Click on *EXAMINE,* rename this REGIONAL BLOCK and set the density to something ridiculous like 10 gm/cc. This block will be used to model the regional in the gravity profile. The greater you set the density, the less structure you need on this block to set the regional. Otherwise the density is of no consequence, so it does not matter that you chose a value that is physically unreasonable. Rename the crustal block TILL UNIT 1, or something, and set the density to whatever was indicated by your terrain density analysis. At this point, you have a model consisting of a till unit (the top of the till conforms to the topography) plus a deep, dense, regional unit, and the gravity predicted from the model should look roughly like the observed gravity. Splitting existing blocks like this is the only way to add to the model. For now, we can leave the model like this.

Move the two points that define the top of the regional block in so that the top of this block is two or three times wider than the survey, and the top is a few hundred m below the surface. There are two ways to approach the regional. If you have a model for the regional from some other analysis, then set the density of the till to 0, and adjust one of the points defining the top of the REGIONAL block up and down. Since the till is 0 density the predicted data will be a straight line. Adjust the REGIONAL block until that straight line has the same slope as the pre-determined regional. The coordinates of the cursor are displayed in the bottom right corner of the screen, so you can calculate a regional slope. If your pre-determined regional is not a straight line then add a few points to the top of the regional block and move them until the predicted regional looks right. The second way of adjusting the regional is to work with an acceptable density for till unit 1, and adjust the regional block until the beginning and end of the observed data and predicted data agree. Since an arbitrary amount can be added to every gravity station, you can fix one point in the predicted data to agree with one point in the observed data. Do this by clicking on the desired point in the data pane, (but not while in zoom mode). A cross will appear on the selected point, and the adjustment will be made. For most purposes, including creating the regional, it is best to select a point near one end of the profile.

The predicted and observed gravity now differ by some small amount. The model must be modified so that the data misfit is as small as the quality of the data allows. As part of this process the estimate of the till density could be refined by looking at details in the profile - the small ups and downs. Off-profile, the model goes into and out of the screen to infinity. In reality, you know that at least the topography is variable N-S, and probably the rest of the model as well. GM-SYS cannot allow the surfaces of the blocks to slope N-S, but you can truncate them so they only extend a short distance off the profile. In the *EXAMINE* menu. Adjust the 2 1/2 D parameters so that the till unit only continues a short amount North and South. How much N-S truncation can be tolerated before the predicted data starts to look different?

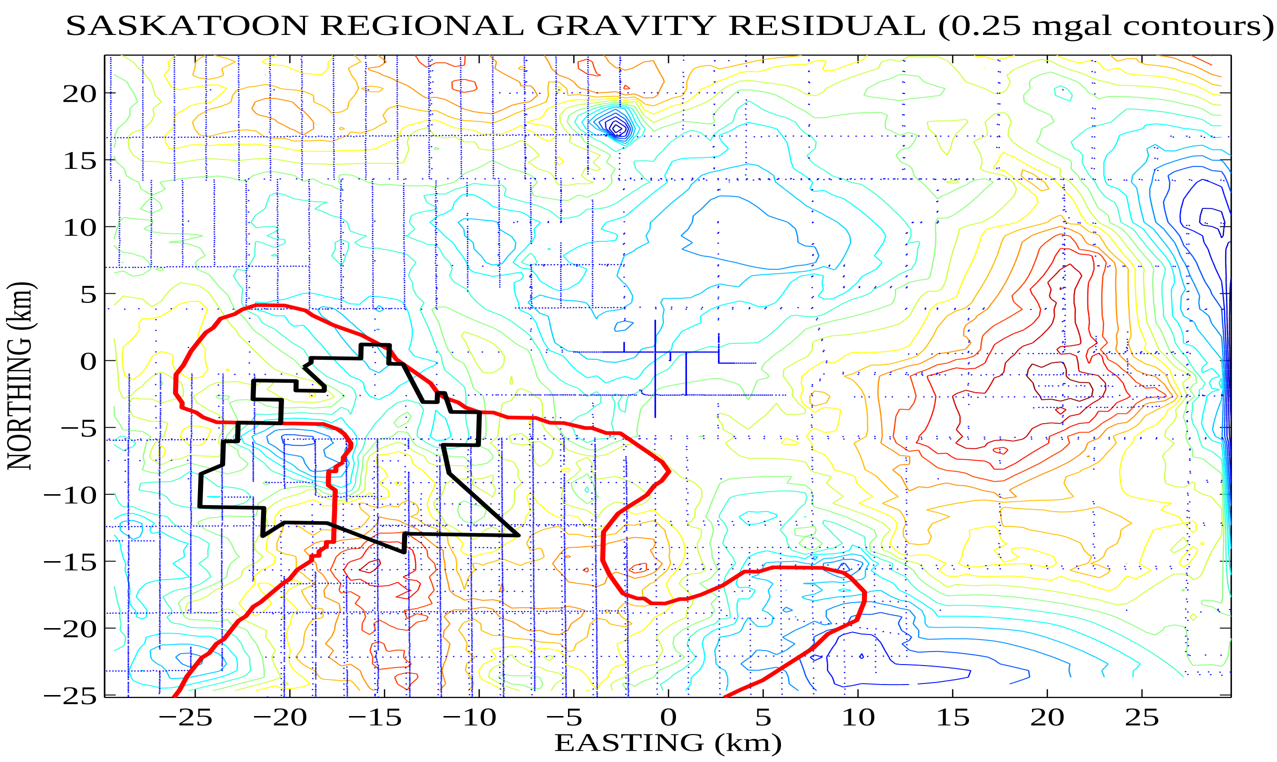
To print your work, click on the printer icon, or from the menu *file, print.* A menu comes up allowing you to select the printer, or file to send the figure to, and a skeleton of the figure. The four horizontal rectangles in the skeleton figure represent the plan view, magnetics, gravity and model panels, relative to the page. Click on the *printer* button to select the printer or type of graphics output. If you select the generic postscript, the block fills in the graphic will be correctly displayed, but you might not be able to import this into CorelDraw, or word and have it scale properly. If you select adobe illustrator, you will get a postscript file, that cannot handle the block fill, but that can be read into CorelDraw and word. After you make his choice select *printer setup*in the main printer menu to set the page size and landscape or portrait. Finally, click on *page layout,* to set the scales for the figure. You can adjust the panel sizes from zero to the whole page, the axis labeling, the tick marks etc in this menu.

Once you get close to an acceptable model you can try to invert the observed data to arrive at a better model. Click on *INV* in the *ACTION* menu to bring up the inversion dialog. Click on *fix/free,* then select *density* and then click on a block in the model panel. The density of this block is now a free parameter. Click *GO* and GM-SYS will search for a change in the density of this block that will result in a smaller data misfit. Click *next step,* and GM-SYS will run through the inversion procedure again to make a further adjustment to the density. You might need to repeat this many times, until further steps result in no further decrease in the data misfit. To change the structure of the model, click on *Z,* and then on several vertices that define the boundary between two blocks, and then *GO*. GM-SYS is making no judgment about the quality or sensibility of the change to the model, only that it results in a smaller data misfit. For this reason you, need to be fairly close to an acceptable model before you try to invert, and you should judge the result of each inversion in terms of its geological sense.

If there are any well logs near the profile, you could display this information on the figure. Click on *profile, edit wells,* and enter the information as appropriate.

1) The field school Bouguer gravity anomaly looks like a contact and one of the critical things to decide is where can that contact be. There are density contrasts at the Precambrian basement, the winnipegosis, the evaporite, the Cretaceous shale Manville and the till shale. Try to model a simple structure at each of these depths and see which can produce the anomaly we observe.

   Solution collapses in the Prairie Evaporite are common features. Saskatoon is near the edge and inside of one. This feature has been explored by numerous boreholes so its extent is fairly well known. The top of the Lea Park shale is depressed by a maximum of 175 m in this area, and that alone should cause a gravity anomaly of 2 mgal if the density contrast between the Lea Park and the infill is 0.2 gm/cc. There is however almost no expression of the collapse (outline in red) in gravity as can be seen from the figure below, so the story is clearly more complicated. The limestones immediately above the evaporite are denser than the evaporite, so when they collapse into the evaporite they create a positive density contrast and hence a positive gravity anomaly. Everywhere else in the section above the evaporite where there is a density contrast, either positive or negative, there should be a contribution to gravity. The sum of all these is evidently smaller than the isolated effect of any one, or the connection with gravity would be more obvious.



2) Construct a GMSYS model of a solution collapse that comes all the way up to the top of the Cretaceous shale, with the shales filled in with till. A simplified stratigraphy suggests that there are large density contrasts between the evaporite and the Dawson Bay limestones, at a depth of 1100 m, the Mannville and the Duperow limestone at say 500 m, Cretaceous shale and Mannville at 300m, and the shale and the till at 100 m. Assume that the collapse is entirely vertical, that the collapse is 100 thick at all

horizons and that the density contrasts are as in the table below.

COLLAPSE MODEL PARAMETERS

UNITS	$\Delta\rho$ gm/cm^3	Depth m
LIMESTONE-SALT	+0.2	1100
SANDSTONE-LIMESTONE	-0.2	500
SHALE-SANDSTONE	-0.2	300
TILL-SHALE	+0.2	100

OASIS MONTAJ AND GMSYS

There are some differences between the stand alone GMSYS we used to use and the new version, which is a part of MONTAJ. First of all a data file looks like

AN EXAMPLE OF A GRAVITY DATA FILE
TO BE READ INTO A MONTAJ GMSYS PROJECT

```
This could be a comment line
  This too
LINE XXXX
    100.0      635.0     574.65     26.32
    150.0     1635.0     575.74     26.43
```

This version can accept up to 1000 lines. Notice that you can have some comment lines and you instruct the program that reads the file that the data starts below the LINE XXXX line, where XXXX can be anything. The first column is the x coordinate, the second column is the y coordinate, the third column is the elevation and the fourth column is gravity. They do not have to be in this order, as the identity of each column can be specified when the file is read.

- Create a MONTAJ Directory in MY DOCUMENTS
- Create a directory in the MONTAJ directory to hold the GMSYS project files - call it for example MYGMSYS
- Place a copy the data files you want to use in the MYGMSYS directory
- Open a DOS COMMAND PROMPT window (this may be in Accessories in the All Programs pull up)
- Traverse to the MYGMSYS folder using the DOS CD command eg CD MONTAJ CD MYGMSYS
- From this folder execute the following command

c:\fs_gms \ xyzgms INFILE.xtn OUTFILE -l=LXXXX -ZT=3 -ZEG=3 -ZG=4

The directory fs_gms is already on those computers and xyzgms is a program that reads the input data and constructs the output data files for the project. If fs_gms is not already a directory under c: then create it and copy xyzgms from (c:\ Program Files(x86) \ Geosoft\ Oasis montaj-Educational\ bin). INFILE.xtn is the name of the data file you want to read in. You need the extension .xtn or .txt or whatever; OUTFILE is the name of the output files that will be created. No extension is required on OUTFILE, as the program generates several files all with their own extension; -l=LXXXX (remember the LINE XXXX near the top of the input file that says where data starts, this is where you identify that line to the program); -ZT=3 means the topo data is in column 3 of the input file; -ZEG=3 means the elevation of the gravity stations is in column 3 (in most cases the topo and the elevation of the stations will be the same, but not always); -ZG=4 means the gravity is in column 4.

- run MONTAJ from ALL PROGRAMS ... GEOSOFT... MONTAJ
- From the FILE pull down menu in MONTAJ create a new MONTAJ project in the MONTAJ folder below MY DOCUMENTS (NOT IN THE MYGMSYS FOLDER).
- In MONTAJ select the GX pull down menu and then select the LOAD MENU and from that select gmsys.omn, which will load the GMSYS software into MONTAJ. Once the MONTAJ project has been saved, this GMSYS project will always be available from within MONTAJ.
- From the GMSYS pull down menu select Existing Model and then Select Open A model. A pop up menu will appear with a space for a filename. Use the ... button to navigate to the folder holding the GMSYS project (ie MYGMSYS below MONTAJ below MY DOCUMENTS), where you should see the file with extension .gms that you want.
- You may get a warning about origin and projection, just click OK.
- The instructions for operating GMSYS are at the front of the lab.

If you save your model before exiting, then when next you want to work with it, start at the first bullet above and select open and existing model.