

User Guide

for Oasis montaj Depth to Basement

Oasis montaj 2023.1

This guide was developed using Oasis montaj version 2023.1. If you are using a different version of Oasis montaj, some features may not be available or may work differently from what is described in this guide. Guides and courses are regularly updated. To see the latest, visit [Seequent Learning Centre](#) and search the catalogue for “Oasis montaj.”

© 2023 Seequent, The Bentley Subsurface Company

All rights reserved. Unauthorised use, reproduction, or disclosure is prohibited. Seequent assumes no responsibility for errors or omissions in this document.

Contents

Getting Started with Depth To Basement	1
Loading the Depth to Basement Menu	1
To Load the Depth to Basement menu	1
Data Requirements for All Depth to Basement Methods	2
References	2
Generating Solutions with Depth to Basement	4
Generating Werner Solutions	4
To Generate Werner Solutions	4
Output Solutions	6
Generating Werner Solutions for Gravity Data	7
Generating Analytic Signal Solutions	9
To Generate Analytic Signal Solutions	9
Output Solutions	10
Generating Analytic Signal Solutions for Gravity Data	11
Generating Extended Euler Solutions	12
To Generate Extended Euler Solutions	12
Output Solutions	14
Generating Extended Euler Solutions for Gravity Data	15
Clustering Solutions	16
To Cluster Solutions	16
Output Solutions	17
Why Some Calculated Susceptibilities are Large	18
Displaying and Exporting Depth to Basement Solutions	19
Displaying Solutions in Oasis montaj	19
To Display Solutions as Coloured Symbols on a Map	19
To Display Solutions as Symbols in a 3D View	22
Exporting Solutions to GM-SYS Profile	23
Export Solutions for Display in GM-SYS Profile	23
Creating GM-SYS Profile Models	24
To Generate GM-SYS Models from the Profile Database	25
Creating GM-SYS Profile Models and Exporting Solutions in One Pass	26

Getting Started with Depth To Basement

1. [Loading the Depth to Basement Menu](#)
2. [Data Requirements for All Depth to Basement Methods](#)
3. [References](#)

This guide describes how to use the Depth to Basement extension to determine the position (distance along the profile and depth), dip (orientation) and intensity (susceptibility) of source bodies from magnetic data that is stored in a Geosoft database. The extension can also be used on gravity data to determine the position of source bodies from gravity data, provided there is a significant contrast in density between the source and its surroundings.

This extension includes three different Profile Depth to Basement (PDepth) techniques; *Werner Deconvolution*, *Analytic Signal* and *Extended Euler Deconvolution*. Each Depth to Basement function uses a different accepted technique for determining the depth to the source. Each method has advantages in particular geologic situations. Applying multiple methods to the same anomaly profile greatly improves the reliability of results.

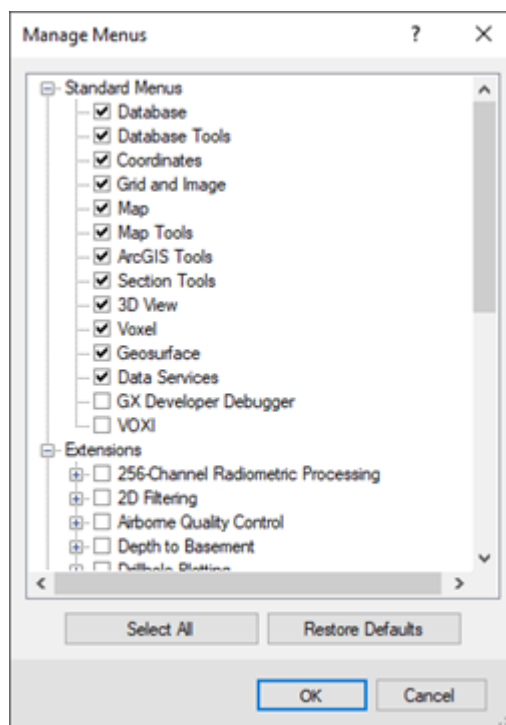
Solutions are saved in a new Geosoft database, enabling you to immediately view the results in profile, edit the solutions, and plot the solutions in maps or 3D views. Additional functions also enable you to cluster solutions, export solutions to GM-SYS models, and generate starting GM-SYS models from data profiles.

Loading the Depth to Basement Menu

Before you can start using the Depth To Basement extension you must load the menu onto the main Oasis montaj menu bar.

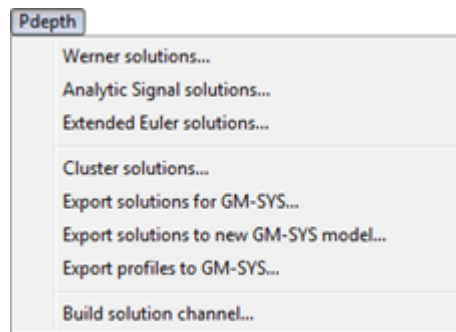
To Load the Depth to Basement menu

1. On the *Project* menu, select **Manage Menus**.
The *Manage Menus* dialog appears.



2. Open the *Extensions* list, check the *Depth to Basement* menu from the list and click **OK**.

The new *Pdepth* menu appears on your menu bar.



Data Requirements for All Depth to Basement Methods

Each method requires X, Y, elevation, topography and anomaly channels for the input database. Topography may be all dummies. The difference between elevation and topography channels is subtracted from the solution depths to generate the channel containing the depth of the solutions relative to sea level, *Depth_sl*. All channels containing distance units are assumed to use the same units.

All methods require the specification of minimum and maximum source depths, in the same distance units, that define the solution space. Solutions shallower than the minimum depth or deeper than the maximum depth will be discarded.

Depth to Basement methods are designed primarily for use with total-field magnetic data. For each method, you must set the magnetic field parameters to those appropriate to the magnetic survey. However, mathematically the magnetic response is equivalent to the derivative of the gravity response, so it is possible to use Depth to Basement methods on gravity profiles by using either the horizontal or vertical derivative of gravity as the input anomaly profile, rather than total-field magnetic data.

- When using the vertical derivative, set the *Inclination* to 90 degrees and the *Declination* to 0.
- For the horizontal derivative, set the *Inclination* to 0, and the *Declination* to the profile azimuth.

Setting the *Field Strength* to 10 seems to give the best conversion of susceptibilities to densities. However, remember that the susceptibilities calculated for dike solutions are really effective susceptibilities (i.e., susceptibility * width). Also remember that the contact solutions are calculated from the 2nd derivative of the gravity profile. Unless the original gravity profile is especially clean, the 2nd derivative will tend to be noisy, so the density contrasts calculated are often not very accurate.

The methods use horizontal and vertical derivatives in the calculations. You may either specify existing derivative channels to use, or have one or more derivatives calculated from the input channel. If your input profiles are noisy, you can improve the performance of Depth to Basement methods significantly by low-pass filtering or smoothing the anomaly profiles prior to running the methods. This is especially true in the case of gravity profiles, where the derivatives are really 2nd derivatives. The calculated susceptibilities and densities will be less accurate with noisy data.

All of the methods assume that the anomaly profile is perpendicular to the geologic structure generating the field. Set the *Relative Strike* parameter to reflect the clockwise difference between the profile azimuth and the geologic strike.

References

- Ku, C.C. and Sharp, J.A., 1983, Werner deconvolution for automated magnetic interpretation and its refinement using Marquart's inverse modeling: *Geophysics*, vol. 48, no. 6, p754-774

- Mushayandebvu, M. F., van Driel, P., Reid, A.B., and Fairhead, J.D., 2001, Magnetic source parameters of two-dimensional structures using extended Euler deconvolution: *Geophysics*, vol. 66, no. 3, p814-823
- Nabighian, M. N., 1972, The Analytic Signal of Two-Dimensional Magnetic Bodies with Polygonal Cross-Section: Its Properties and Use for Automated Anomaly Interpretation: *Geophysics*, vol. 37, no. 3, p507-517
- Nabighian, M. N., 1974, Additional Comments on the Analytic Signal of Two-Dimensional Magnetic Bodies with Polygonal Cross-Section: *Geophysics*, vol. 39, no. 1, p85-92
- Phillips, J. D., 1997, Potential-Field Geophysical Software for the PC, version 2.2: USGS open-File Report 97-725
- Reid, A.B., Allsop, J.M., Granser, H., Millett, A.J., and Somerton, I.W., 1990, Magnetic interpretation in three dimensions using Euler deconvolution: *Geophysics*, vol 55, no 1, p80-91

Generating Solutions with Depth to Basement

The chosen Depth to Basement method will operate on all selected lines in the current database. Simply select one of the methods to use. Adjust the method-specific parameters to control the number of solutions generated by the calculations.

Depth to Basement methods recreate the output database each time a method is run, so previous solutions may be overwritten.

1. [Generating Werner Solutions](#)
2. [Generating Analytic Signal Solutions](#)
3. [Generating Extended Euler Solutions](#)
4. [Clustering Solutions](#)

Generating Werner Solutions

This technique is based on the USGS program PDEPTH (Phillips, 1997) and includes the iterative improvement scheme described by Ku and Sharp (1983). The input database lines are interpolated to an even sample interval using the standard Oasis montaj spline prior to Werner deconvolution. The sample interval is the total profile length divided by the number of points in the line. Lines with large gaps should be split into separate lines.

Each Werner deconvolution calculation operates on a segment of a selected database line, referred to as a *window*, and may produce a single solution. Starting at the beginning of the profile using the smallest window size, the window moves along the profile to the end. Then the window size is increased by a specified increment and the selected database line is processed again.

For noisy input data, results can be improved significantly by filtering the input anomaly and gradient data. Werner uses a cubic spline technique to calculate the horizontal derivative if the user does not specify an input gradient channel.

To Generate Werner Solutions

1. Ensure that the database you want to analyze is open and selected in your current project.
2. From the *Pdepth* menu, select **Werner solutions**.
The *Werner - generate Werner solutions* dialog appears.

Werner - generate Werner solutions

X Channel: X

Y Channel: Y

Elev. Channel:

Mag Channel:

Horizontal Derivative: <Calculate>

Topography: <None>

Min. Depth: 1

Max. Depth: 5000

Min. Window Length:

Max. Window Length:

Window Expansion Increment:

Window Shift Increment:

Detrend Order: 1

Relative Strike: 90.0

Field Strength: 50000.0

Inclination: 60.0

Declination: 20.0

Residual cut-off: 0.0

X cut-off: 20.0

Output Database Name: ...

OK Cancel

3. Using the dropdown lists, select the *X Channel*, *Y Channel*, *Elev. Channel* and *Topography* (Note that, the distance units must be the same for all four parameters and the Elevation and Topography are positive up, relative to sea level).
4. Use the *Mag. Channel* dropdown list to select the magnetic (or gravity) channel.
5. Use the *Horizontal Derivative* dropdown list to select the input horizontal derivative channel. Choosing the **<Calculate>** option will calculate the derivative from the input *Mag Channel* values.
6. Specify the *Min. Depth* and the *Max. Depth*. Solutions shallower or deeper are discarded. (Note that, minimum and maximum depth are relative to the survey elevation, i.e., positive down).
7. Several parameters in the dialog control the number of solutions generated by Werner:
 - *Min. Window Length* sets the minimum lengths of the Werner operator.
 - *Max. Window Length* set the maximum lengths of the Werner operator.
 - *Window Expansion Increment* determines the number and size of steps between the minimum and maximum.
 - *Window Shift Increment* sets the distance the Werner operator is moved along the profile between calculations.

These four parameters are specified in the distance units of the input database. Smaller values of *Window Expansion Increment* and *Window Shift Increment* generate more calculations and hence more solutions.

8. You may elect to have a trend removed from the data in each window as part of the calculation. In the *Detrend Order* parameter you can specify 0-order, 1st-order, or 2nd-order trend removal.
9. Specify the *Relative Strike*, which is the angle between profile direction and anomaly strike in degrees, positive counterclockwise. The default 90 degrees.
10. Specify the *Field Strength*, which is used to calculate susceptibility, which will be in same units.
11. Specify the *Inclination* and *Declination* in degrees.

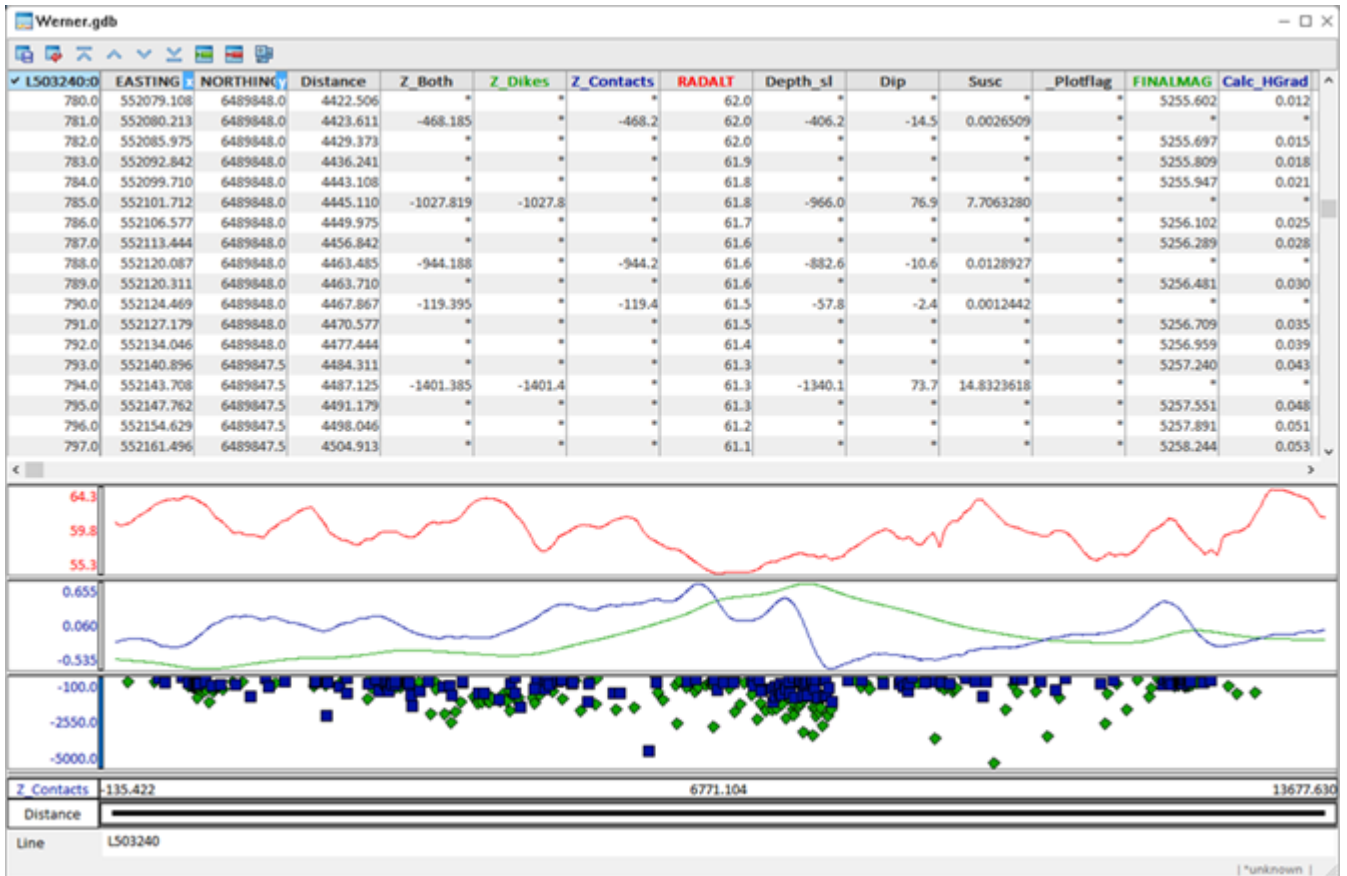
Each Werner calculation potentially generates one solution. Two input parameters determine whether a calculated solution will be saved in the output database: *Residual cut-off* sets an amplitude threshold for anomalies (in nT units) and enables you to eliminate solutions caused by noise in the input profile. Larger values of *Residual cut-off* eliminate more solutions. *X cut-off* sets a horizontal distance threshold (in fiducials units) for solutions relative to the center of the Werner operator. When *X cut-off* is decreased, more solutions are eliminated. Values of *X cut-off* greater than 2, combined with small values of *Window Shift Increment*, generate the spray patterns seen in some examples in the literature.

12. Specify the *Output Database Name* in the text box provided.

The Werner Deconvolution method will not find many valid solutions at depths shallower than the input data spacing or deeper than the window length.

Output Solutions

Results are written to the specified output database, which should look similar to what is shown below.



The solutions are sorted by distance along the line, referenced to the first point in the input line. The channel *Z_Dikes* contains the dike solution depths relative to the flight elevation. The channel *Z_Contacts* contains the contact solution depths relative to the flight elevation. All of the solutions are also in a 3rd channel named *Z_Both*. The channel *Depth_sl* contains the depth relative to sea level calculated using the input *Elev.* channel if it was populated. The sign of the Z-axis is negative down for all of these channels to enable profiles to be plotted. A hidden channel *Dike0_Cont1* contains a flag identifying the solution as a dike (0) or a contact (1).

Four of the specified input channels, re-sampled to an even sample interval are copied to the output database: *Elev.*, *Mag*, *Horizontal Gradient*, and *Topography*. *Elev* and *Topography* profiles are automatically plotted in the top pane; the *Mag* and *Horizontal Gradient* profiles are automatically plotted in the middle pane; and symbols for the *Z_dikes* and *Z_Contacts* are automatically plotted in the bottom pane.

Generating Werner Solutions for Gravity Data

To use Werner Deconvolution on gravity data, use the first vertical derivative of gravity as the input channel rather than total-field magnetics.

If the *Field Strength* is set to 1, *Inclination* set to 90, and the *Declination* set to zero, the *Suse* output channel will be the calculated density contrast. Note that the Contact solutions are computed from the 2nd horizontal derivative in the gravity case, so some low-pass filtering is often required.

Generating Analytic Signal Solutions

You can use the Analytic Signal Method to calculate the analytic signal depth solutions from magnetic (or gravity) profiles. This method is based on the USGS program PDEPTH (Phillips, 1997), which is based on the method published by Nabighian (1972, 1974). The input profiles are interpolated to an even sample interval using the Oasis montaj spline before processing. The sample interval is the total profile length divided by the number of points in the profile. Profiles with large gaps should be split into multiple lines.

For noisy input profiles, the results can be improved significantly by filtering the input anomaly and gradient data. The horizontal derivative is calculated via FFT if the user does not specify an input gradient channel.

When using the Analytic Signal method on gravity profiles, use the horizontal derivative of gravity as the input profile, rather than total-field magnetics. If the *Field Strength* is set to 1, *Inclination* set to 90, and the *Declination* set to zero, the *Susc* output channel will be the calculated density contrast.

The "Contact" solutions are computed from the 2nd horizontal derivative in the gravity case, so some low-pass filtering is often required.

To Generate Analytic Signal Solutions

1. From the *Pdepth* menu, select **Analytic Signal solutions**.
The *Anasig - generate Hilbert solutions* dialog appears.

The screenshot shows the 'Anasig - generate Hilbert solutions' dialog box. It has a title bar with a question mark and a close button. The dialog contains the following fields and controls:

- X Channel:** EASTING (dropdown)
- Y Channel:** NORTHING (dropdown)
- Elev. Channel:** (empty dropdown)
- Mag Channel:** (empty dropdown)
- Horizontal Derivative:** <Calculate> (dropdown)
- Topography:** <None> (dropdown)
- Min. Depth:** 1 (text input)
- Max. Depth:** 5000 (text input)
- Min. Window Length:** (empty text input)
- Max. Window Length:** (empty text input)
- Window Expansion Increment:** (empty text input)
- Relative Strike:** 90.0 (text input)
- Field Strength:** 50000.0 (text input)
- Inclination:** 60.0 (text input)
- Declination:** 20.0 (text input)
- Output Database Name:** (empty text input with a browse button '...')

At the bottom right, there are 'OK' and 'Cancel' buttons.

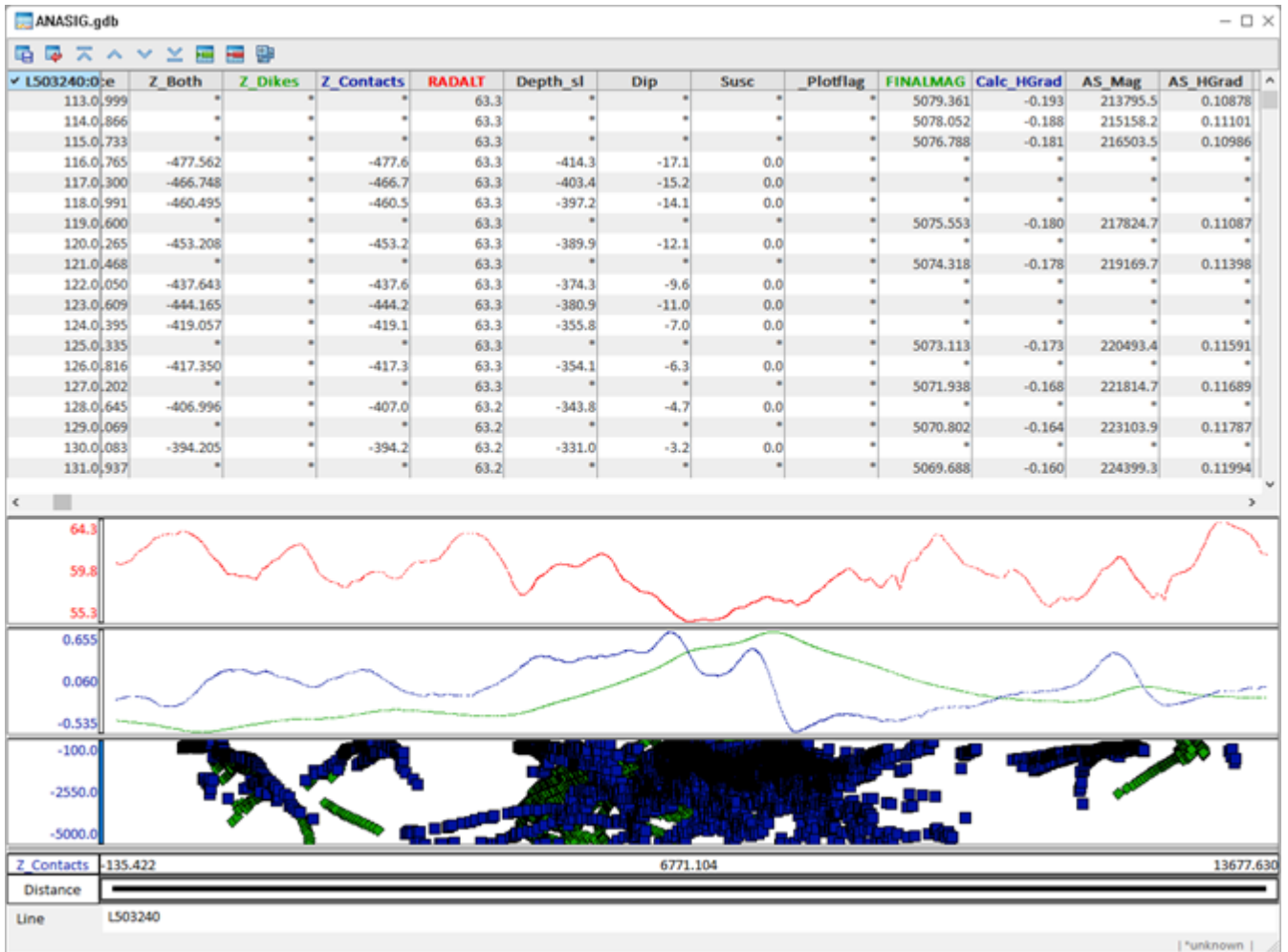
2. Using the dropdown lists, select the *X Channel*, *Y Channel*, *Elev. Channel* and *Topography* (Note that, the distance units must be the same for all four parameters and the Elevation and Topography are positive up, relative to sea level).
3. Use the *Mag. Channel* dropdown list to select the magnetic (or gravity) channel..
4. Use the *Horizontal Derivative* dropdown list to select the input horizontal derivative channel.
5. Specify the *Min. Depth* and the *Max. Depth*. Solutions shallower or deeper are discarded. (Note that, minimum and maximum depth are relative to flight elevation, positive down).
6. Three parameters in the dialog control the number of solutions generated by the Analytic Signal (Anasig) method.
 - *Min. Window Length* sets the minimum lengths of the analytic signal operator.
 - *Max. Window Length* set the maximum lengths of the analytic signal operator.
 - *Window Expansion Increment* determines the number and size of steps between the minimum and maximum.

These three parameters are specified in the distance units of the input database. Smaller values of *Window Expansion Increment* and *Window Shift Increment* generate more calculations and hence more solutions.

7. Each analytic signal calculation may generate a single solution. After a local peak is detected in the anomaly profile, operators that vary from *Min. Window Length* to *Max. Window Length* are used to calculate an interpolated location for the peak and a depth.
8. Specify the *Relative Strike*, which is the angle between profile direction and anomaly strike in degrees, positive counterclockwise. The default 90 degrees.
9. Specify the *Field Strength*, which is used to calculate susceptibility, which will be in same units.
10. Specify the *Inclination* and *Declination* in degrees.
11. Specify the *Output Database Name* in the text box provided.
 - If the *Field Strength* is set to 1, *Inclination* set to 90, and the *Declination* set to zero the "Susc" output channel will be the calculated density contrast.

Output Solutions

Results are written to the specified output database, which should look similar to what is shown below.



The solutions are sorted by distance along the line, referenced to the first point in the input line. The channel *Z_Dikes* contains the dike solution depths relative to the flight elevation. The channel *Z_Contacts* contains the contact solution depths relative to the flight elevation. All of the solutions are also in a 3rd channel named *Z_Both*. The channel *Depth_sl* contains the depth relative to sea level calculated using the input *Elev*. channel if it was populated. The sign of the Z-axis is negative down for all of these channels to enable profiles to be plotted. A hidden channel *Dike0_Cont1* contains a flag identifying the solution as a dike (0) or a contact (1).

Four of the specified input channels, re-sampled to an even sample interval are copied to the output database: *Elev.*, *Mag*, *Horizontal Gradient*, and *Topography*. *Elev* and *Topography* profiles are automatically plotted in the top pane; the *Mag* and *Horizontal Gradient* profiles are automatically plotted in the middle pane; and symbols for the *Z_dikes* and *Z_Contacts* are automatically plotted in the bottom pane. The calculated Analytic Signal of the input total-field magnetics is saved in the database as *AS_Mag* and the calculated Analytic Signal of the input horizontal-gradient magnetics is saved in the database as *AS_HGrad*.

Generating Analytic Signal Solutions for Gravity Data

To use Analytic Signal Deconvolution on gravity data, use the vertical derivative of gravity as the input data rather than total-field magnetics. If the *Field Strength* is set to 1, *Inclination* set to 90, and the *Declination* set to zero, the *Susc* output channel will be the calculated density contrast. Note that the Contact solutions are computed from the 2nd horizontal derivative in the gravity case, so some low-pass filtering is often required.

Generating Extended Euler Solutions

The Extended Euler algorithm used in this method was provided by GETECH and is based on the paper from Mushayandebvu et. al. (2001). This approach calculates solutions using both the conventional Euler equation (Reid et. al., 1990) and the rotational constraint equation from Extended Euler. Solving both equations jointly (Extended Euler) gives distance, depth, dip, and susceptibility, assuming there is no remanent magnetization. Conventional Euler Deconvolution is also used to provide a second estimate for distance and depth. If the relative difference in depth for the two estimates is less than the maximum percentage error given by the user, the solution is retained; otherwise it is rejected.

The input profiles are interpolated to an even sample interval using the Oasis montaj spline before processing. The sample interval is the total profile length divided by the number of points in the profile. Therefore, profiles with large gaps should be split into multiple lines.

This method uses an FFT technique to calculate the horizontal and vertical derivatives if the user does not specify an input gradient channels. For noisy input profiles, the results can be improved significantly by filtering the input anomaly and gradient data.

To Generate Extended Euler Solutions

1. Ensure that the database you want to analyze is open and selected in your current project.
2. From the *Pdepth* menu, select **Extended Euler solutions**.
The *ExEuler - generate Extended Euler solutions* dialog appears.

The dialog box 'ExEuler - generate Extended Euler solutions' contains the following fields and options:

- X Channel:** EASTING (dropdown)
- Y Channel:** NORTHING (dropdown)
- Elev. Channel:** (empty dropdown)
- Mag Channel:** (empty dropdown)
- Horizontal Derivative:** <Calculate> (dropdown)
- Vert. Derivative:** <Calculate> (dropdown)
- Topography:** <None> (dropdown)
- Min. Depth:** 1 (text input)
- Max. Depth:** 5000 (text input)
- Window Length:** (empty text input)
- Max % error:** 10.0 (text input)
- Structural Index:** 1.0 (text input)
- Relative Strike:** 90.0 (text input)
- Field Strength:** 50000.0 (text input)
- Inclination:** 60.0 (text input)
- Declination:** 20.0 (text input)
- Output Database Name:** (empty text input with a browse button '...')

Buttons: OK, Cancel

3. Using the dropdown lists, select the *X Channel*, *Y Channel*, *Elev. Channel* and *Topography* (Note that, the distance units must be the same for all four parameters and the Elevation and Topography are positive up, relative to sea level).
4. Use the *Mag. Channel* dropdown list to select the magnetic (or gravity) channel.
5. Use the *Horizontal Derivative* dropdown list to select the input horizontal derivative channel.
6. Use the *Vertical Derivative* dropdown list to select the input horizontal derivative channel. The key word "*Calculate*" forces Extended Euler to calculate the derivative from the Mag Channel values.
7. Four parameters in the dialog control the number of solutions generated by the Extended Euler operator:
 - *Min. Depth* sets the minimum depth cut-off.
 - *Max. Depth* sets the maximum depth cut-off.
 - *Window Length* sets the length of the ExEuler operator, which is moved across the profile and is used for each calculation.
 - *Max % error* is used to filter out solutions that differ in depth by more than this stated percentage when calculated by both Euler and Extended Euler calculations.

The minimum and maximum depths, as well as the window length parameter are specified in distance units. The distance units in the input parameters are always the same as those for the X and Y input channels. For example, if the X and Y channels are in meters, all the distance, depth inputs and outputs will be in meters.

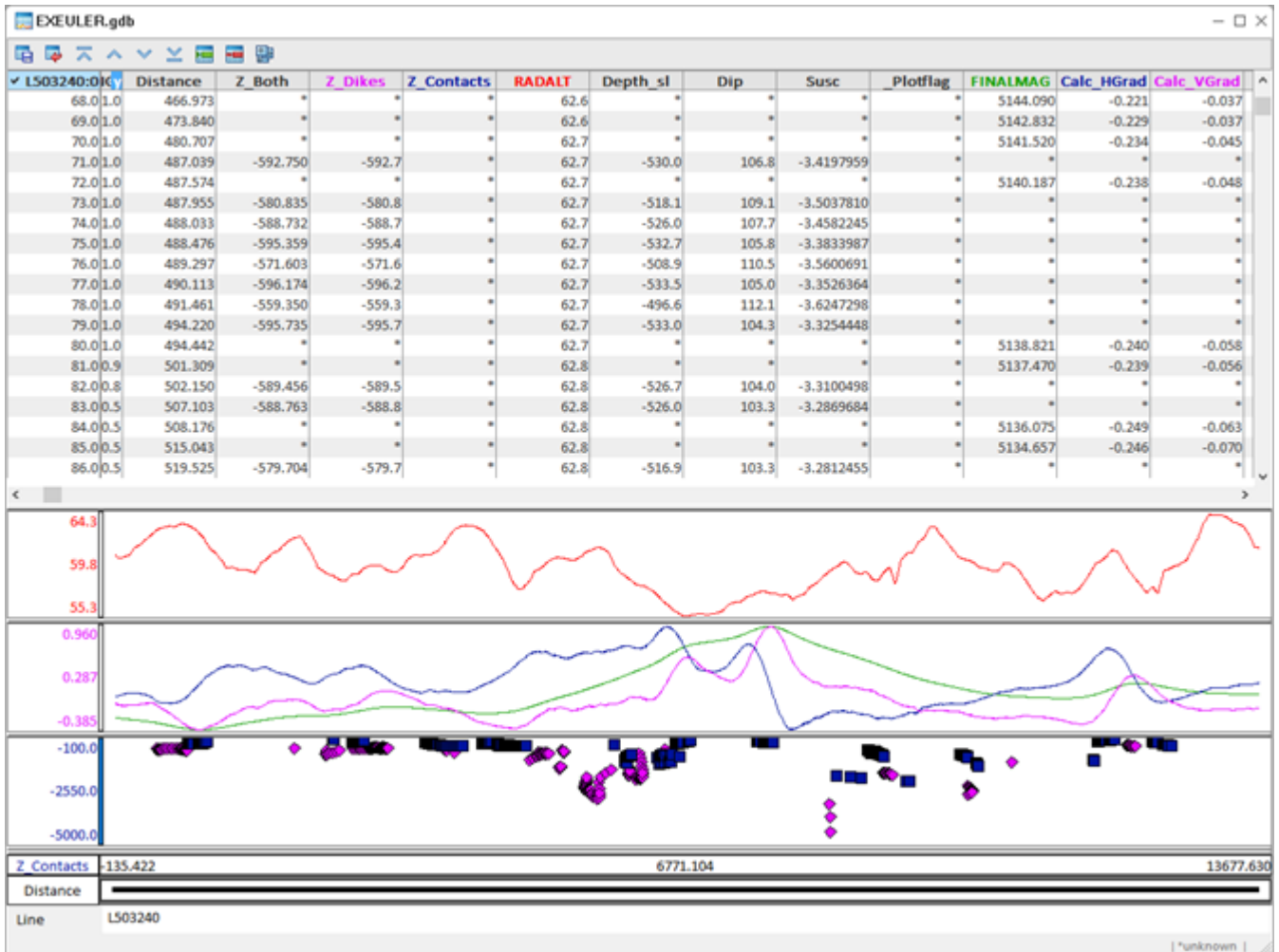
8. Specify the "Dike" *Structural Index*, which is used to calculate solutions output to the *Dike* channel.

The Extended Euler method will not find many valid solutions at depths shallower than the input data spacing or deeper than the window length. Extended Euler always makes two passes through the data. The first pass always uses a Structural Index (SI) of zero to calculate contact solutions. The second pass uses the SI given by the user in the dialog entry for Dike Structural Index. The results of the second pass are always flagged as dike solutions regardless of the SI used.

9. Specify the *Relative Strike*, which is the angle between profile direction and anomaly strike in degrees, positive counterclockwise. The default 90 degrees.
10. Specify the *Field Strength*, which is used to calculate susceptibility, which will be in same units.
11. Specify the *Inclination* and *Declination* in degrees.
12. Specify the *Output Database Name* in the text box provided.

Output Solutions

Results are written to the specified output database, which should look similar to what is shown below.



The solutions are sorted by distance along the line, referenced to the first point in the input line. The channel *Z_Dikes* contains the dike solution depths relative to the flight elevation. The channel *Z_Contacts* contains the contact solution depths relative to the flight elevation. All of the solutions are also in a 3rd channel named *Z_Both*. The channel *Depth_sl* contains the depth relative to sea level calculated using the input *Elev.* channel if it was populated. The sign of the Z-axis is negative down for all of these channels to enable profiles to be plotted. A hidden channel *Dike0_Cont1* contains a flag identifying the solution as a dike (0) or a contact (1).

Five of the specified input channels, re-sampled to an even sample interval are copied to the output database: *Elev.*, *Mag*, *Horizontal Gradient*, *Vertical Gradient* and *Topography*. *Elev* and *Topography* profiles are automatically plotted in the top pane; the *Mag*, *Horizontal Gradient* and *Vertical Gradient* profiles are automatically plotted in the middle pane; and symbols for the *Z_dikes* and *Z_Contacts* are automatically plotted in the bottom pane.

Generating Extended Euler Solutions for Gravity Data

To use Extended Euler Deconvolution on gravity data, use the vertical derivative of gravity as the input data rather than total-field magnetics. If the *Field Strength* is set to 1, *Inclination* set to 90, and the *Declination* set to zero, the *Susc* output channel will be the calculated density contrast. Note that the Contact solutions are computed from the 2nd horizontal derivative in the gravity case, so some low-pass filtering is often required.

Clustering Solutions

The Depth to Basement methods usually generate a spray of solutions around the real solution due to multiple passes with different window lengths. It is often useful to collapse these groups of solutions into single, average solutions for plotting on maps or in cross sections.

To Cluster Solutions

1. From the *PDepth* menu, select **Cluster solutions**.
The *Cluster Depth Solutions* dialog appears.

2. The *Input depth channel* needs to be specified because there are two choices:
 - *Depth_sl* - the depth referenced to sea level
 - *Z_both* - the depth relative to the flight elevation

If you plan to export the solutions to GM-SYS, we suggest that you use "*Depth_sl*".

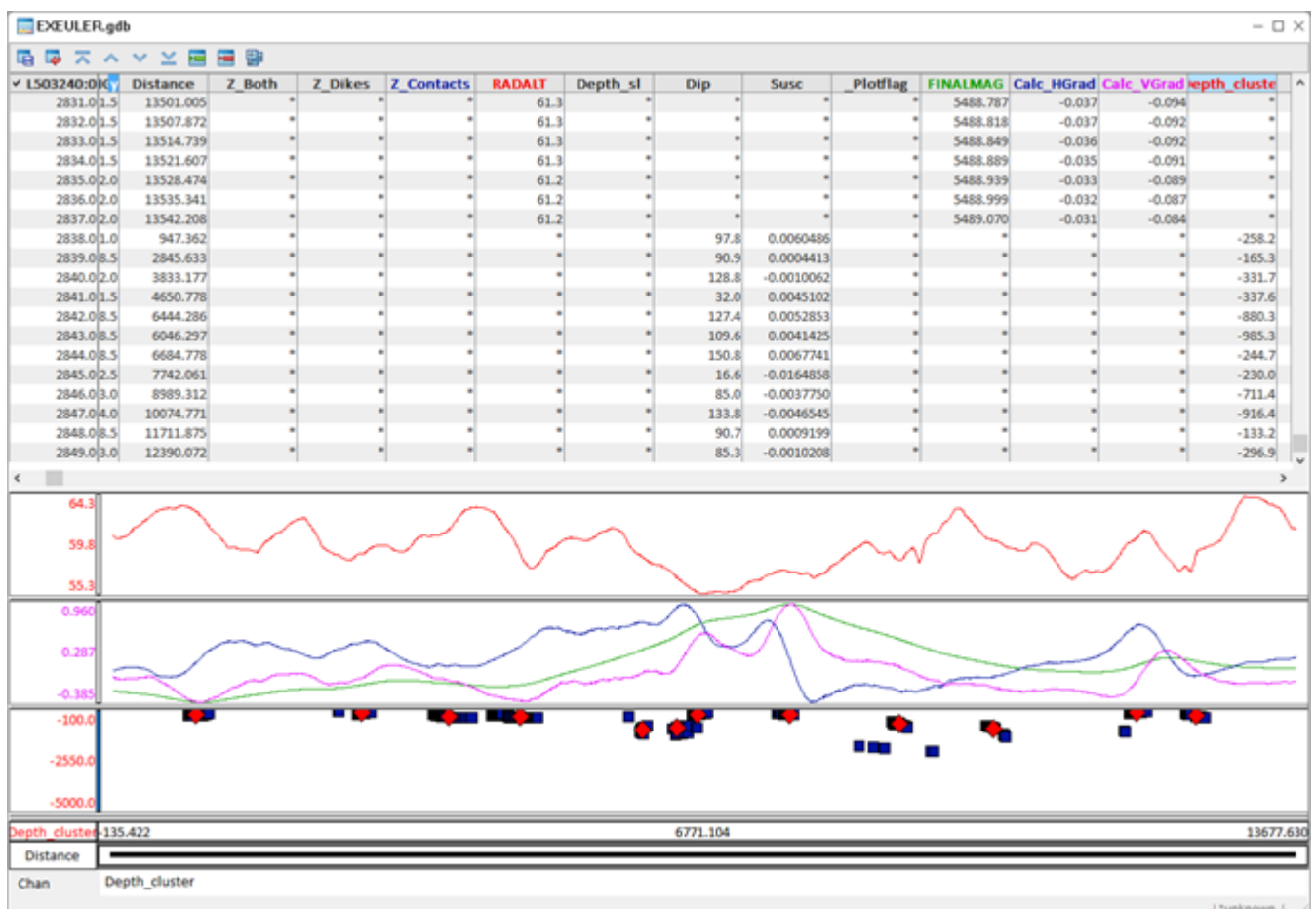
3. Specify the *Output depth channel*, which is the channel in which the clustered solutions depth values are populated.
4. Using the *Input type* dropdown list, select the type of depth solutions to be clustered, that is, enter "0" to cluster dike solutions or "1" to cluster contact solutions.
5. The clustered solutions are appended to the end of each selected line in the current database. The *Output Flag* parameter sets a new flag value for the clustered solutions. Existing solutions with the new flag value are deleted before the new clusters are generated. Assigning different output flag values can save more than one cluster run.
6. Set the *Minimum # of solutions* parameter to indicate how many solutions must be present within the window to generate a clustered solution.

7. Dike solutions and contact solutions must be clustered in separate runs. The user specifies a rectangular window around a solution by specifying the *Window half-width in X* and *Window half-height in Z*.
8. The *Window type* parameter specifies whether these parameters are given in absolute distance units (e.g., meters or kilometres) or as a percentage of depth. The percentage of depth mode provides a mechanism for expanding the window for deeper solutions.
9. If you wish to plot the clustered solutions automatically in the database profiles, you may select **Yes** for the *Plot Clustered Solutions* parameter.
10. You may also select the plotted *Symbol Size* parameter.
11. The number of solutions collapsed to form each cluster is saved in the *Window_Width* channel.

You can work with any distance unit (e.g., meters, kilometres, feet, etc.) as long as you are consistent. The X, Y, distance, and depth channels need to be in the same units.

Output Solutions

Clustered solutions are appended to the bottom of the input database and plotted in the solution window, as shown below, where the clustered solutions (for contacts in this case) are plotted as red diamond symbols.



Why Some Calculated Susceptibilities are Large

If the *Field Strength* input is in nT units, the calculated susceptibilities for contact sources will be in cgs units. However, the susceptibilities calculated for the dike sources are effective susceptibility which is the cgs susceptibility multiplied by the dike width (in the input x,y length units).

For example, if you build a GM-SYS Profile model of a dike that is 100 meters wide, the effective susceptibility calculated by ExEuler will be 100x the actual susceptibility if the distance units are in meters and 0.1x the actual susceptibility if the distance units are in kilometres.

The interpreter can use the large susceptibility values for dike solutions in conjunction with the susceptibilities of nearby contact solutions as an indication of the width of dike-like features. Unfortunately, using SI values for the Field Strength does not yield correct values for susceptibilities.

Displaying and Exporting Depth to Basement Solutions

The *Depth to Basement* extension enables you to display your results in Oasis montaj and export them to the GM-SYS Profile Modeling extension. The following section will guide you through the process for each.

1. [Displaying Results in Oasis montaj](#)
2. [Exporting Results to GM-SYS Profile](#)

Displaying Solutions in Oasis montaj

We have seen in previous sections how solutions are presented in a database in a very similar way for each of the depth estimation methods. These solutions can be plotted as symbols on maps and in 3D views.

To Display Solutions as Coloured Symbols on a Map

1. Select the map you would like to display the solutions on.
2. Select the database that you want to display the solutions from.
3. Open the *Map Tools* menu, hover over the *Symbols* option and select **Colour Range Symbols**. The *Colour Range Symbols* dialog opens.

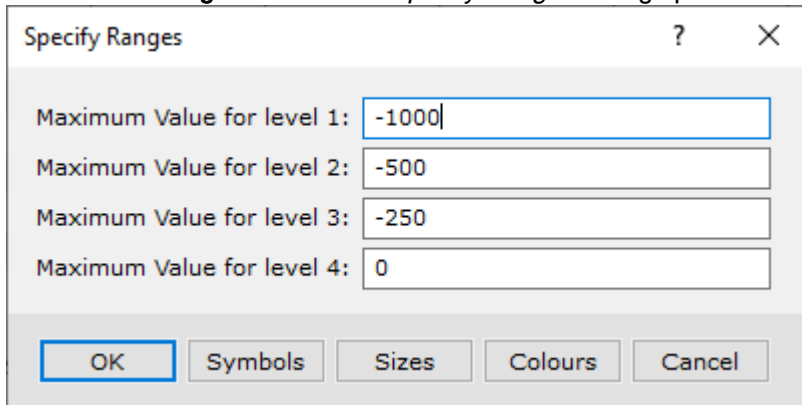
The screenshot shows the 'Colour Range Symbols' dialog box. The settings are as follows:

- Classification channel: Z_Contacts
- Number of ranges: 5
- Mask channel: (empty)
- Symbol font name: GFN Symbols (symbols)
- Symbol weight: medium
- Symbol Edge Colour: Black
- Symbol angle: 0.0

The 'Plot' button is highlighted with a blue border.

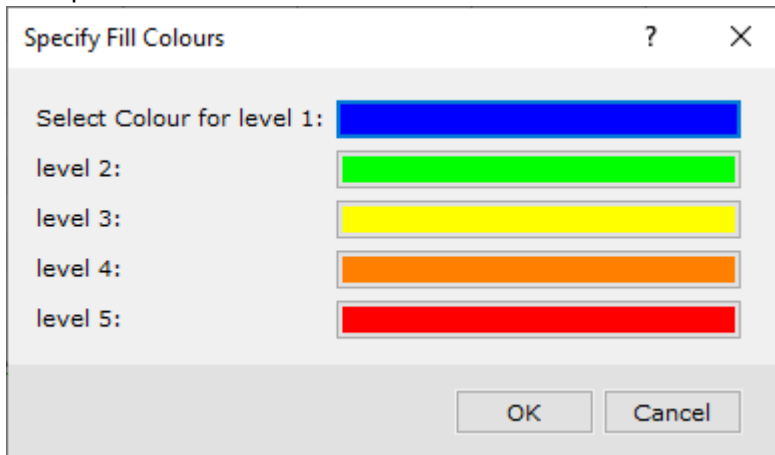
4. Select the *Classification (solution) channel* from the list.
5. Select the *Number of (colour) ranges*.

6. Click on the **Ranges** button. The *Specify Ranges* dialog opens.

The 'Specify Ranges' dialog box has a title bar with a question mark and a close button. It contains four text input fields for maximum values at different levels. The first field is highlighted with a blue border and contains the value '-1000'. The other fields contain '-500', '-250', and '0'. At the bottom, there are five buttons: 'OK' (highlighted with a blue border), 'Symbols', 'Sizes', 'Colours', and 'Cancel'.

Level	Maximum Value
Maximum Value for level 1:	-1000
Maximum Value for level 2:	-500
Maximum Value for level 3:	-250
Maximum Value for level 4:	0

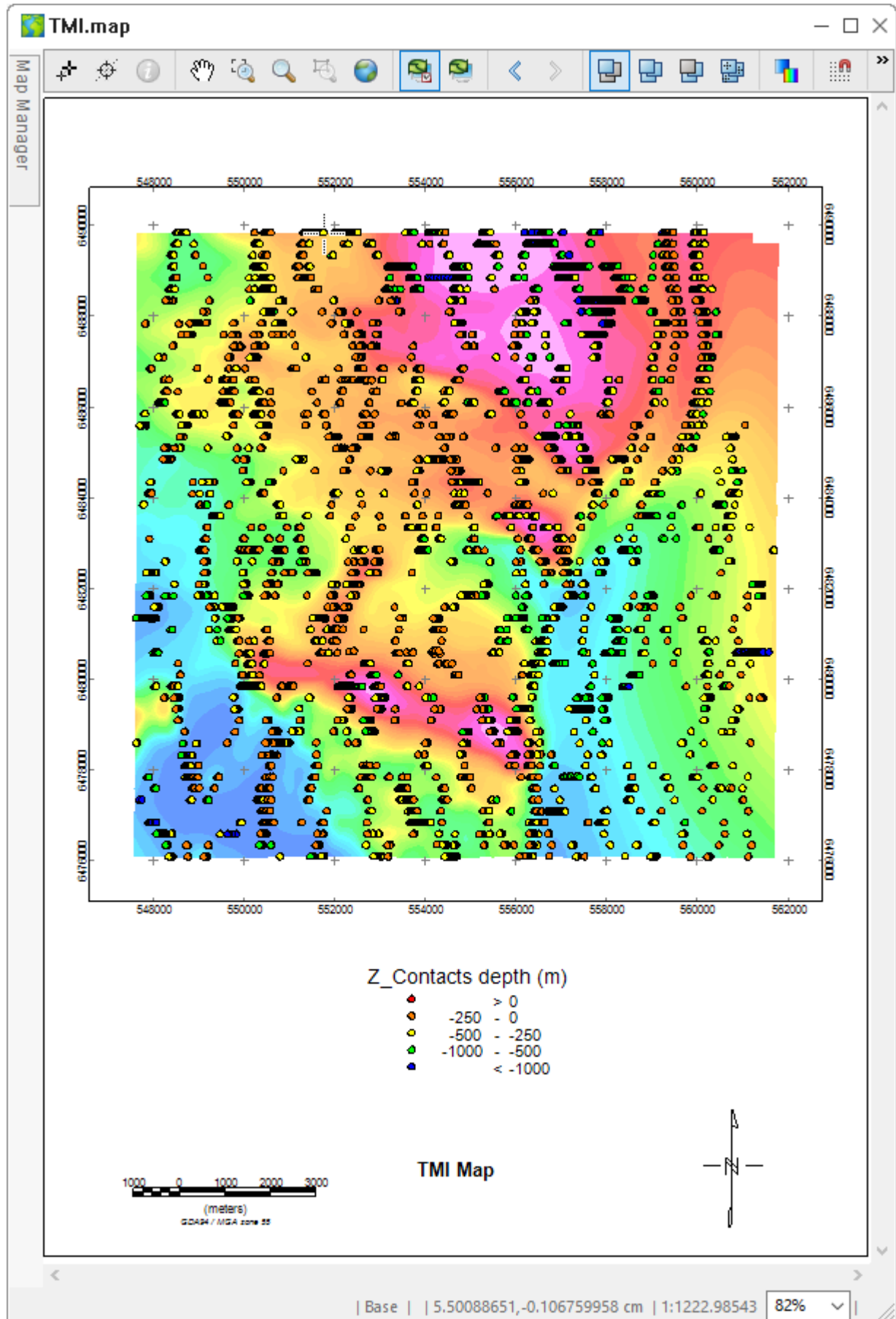
7. Enter the desired depth intervals. The *Symbol* types, *Sizes* and *Colours* can then be specified for each range by clicking on the appropriate button. The *Specify Fill Colours* dialog is shown as an example.

The 'Specify Fill Colours' dialog box has a title bar with a question mark and a close button. It contains five color selection bars for levels 1 through 5. Level 1 is blue, level 2 is green, level 3 is yellow, level 4 is orange, and level 5 is red. At the bottom, there are two buttons: 'OK' and 'Cancel'.

Level	Color
Select Colour for level 1:	Blue
level 2:	Green
level 3:	Yellow
level 4:	Orange
level 5:	Red

8. When finished setting up the symbol display parameters, navigate back to the *Colour Range Symbols* dialog and click **Plot**.

The symbols will plot to the map; an example is shown below.



The colour symbol legend can be created by opening the *Map Tools* menu, hovering over the *Symbols* option and selecting the **Colour Range Symbol Legend** option.

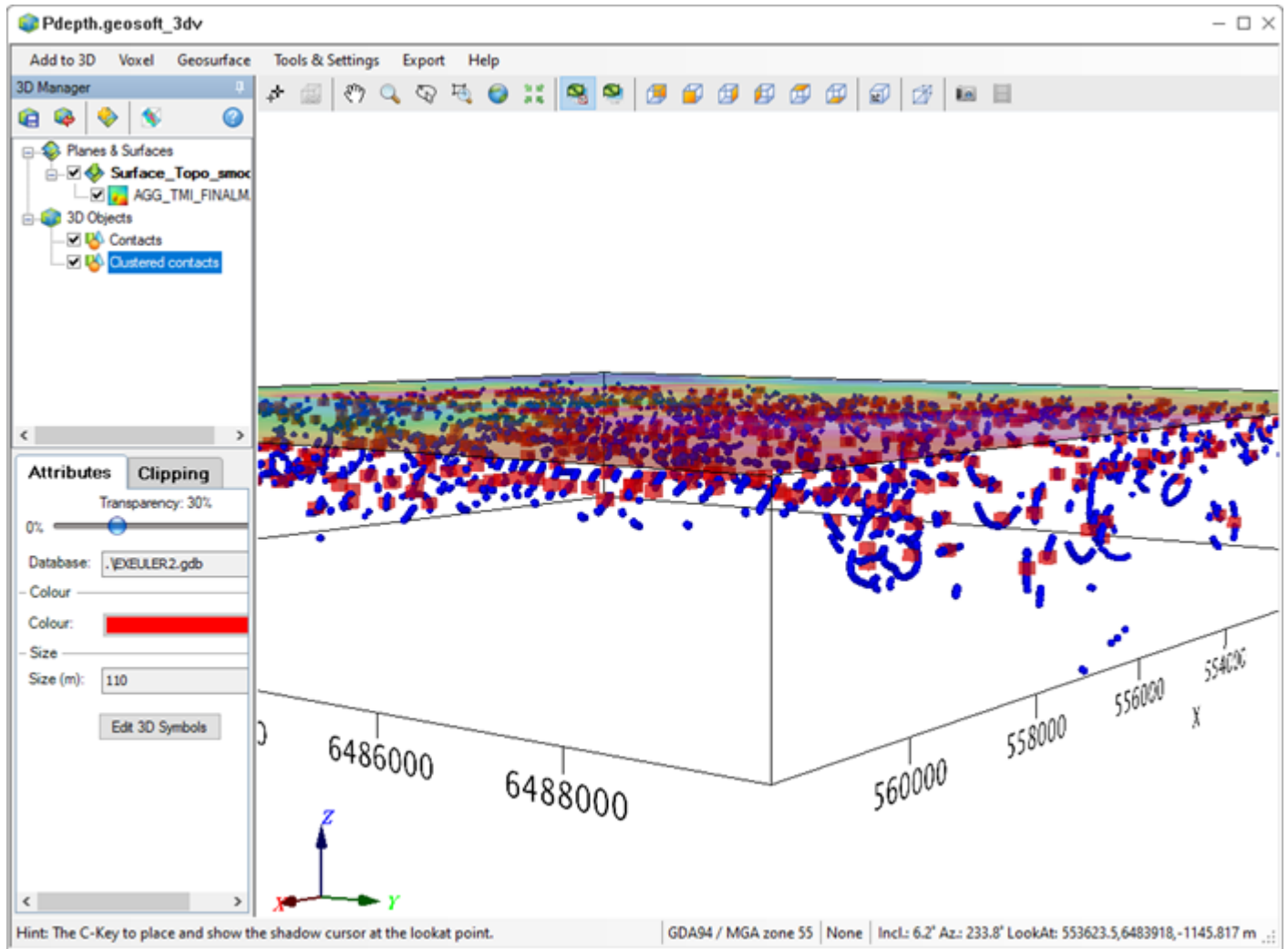
To Display Solutions as Symbols in a 3D View

1. Select the database that you want to plot solutions from.
2. In your 3D view, open the *Add to 3D* menu and select the **3D symbols** option. The *Display 3D Symbols* dialog opens.

The screenshot shows the 'Display 3D Symbols' dialog box. It has a title bar with a question mark and a close button. The dialog is divided into sections for 'Elevation channel', 'Mask channel', 'Shape', 'Colour', and 'Size'. The 'Elevation channel' is set to 'Contacts_Depth'. The 'Mask channel' is empty. The 'Shape' is set to 'sphere'. The 'Colour' section has a radio button for 'Fixed' selected and a blue color bar. The 'Size' section has a radio button for 'Fixed' selected and a text box for 'Size (m)' set to '70'. There are 'OK' and 'Cancel' buttons at the bottom right.

3. Specify the (solution) *Elevation channel* from the database.
4. Specify the desired *Shape*, *Colour* and *Size*. Symbols can be variable sized and coloured by elevation, if desired.
5. Click OK to plot the solutions.

An example is shown below, where contact solutions have been displayed as blue spheres and the clustered contact solutions have been displayed as red cubes.



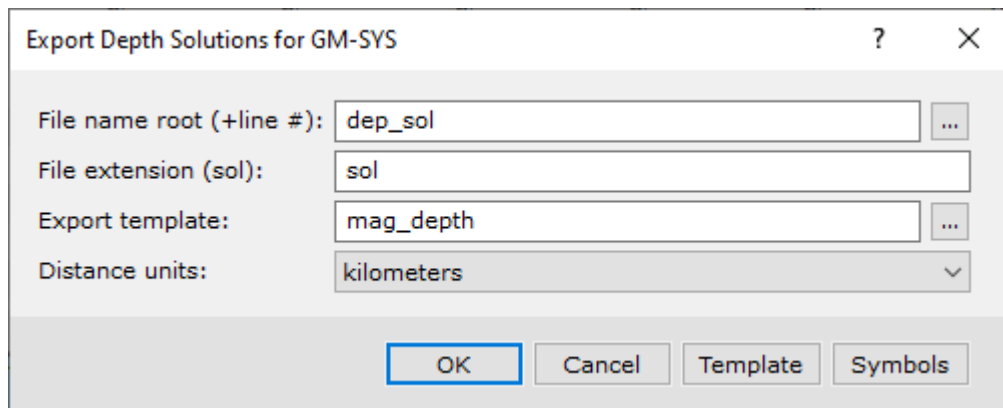
Exporting Solutions to GM-SYS Profile

GM-SYS Profile Modeling is an extension that allows you to build a 2D geological model along a profile, while forward calculating the gravity and magnetic response of the model. Depth solutions from Pdepth can be very useful guides in the creation and/or validation of these models. Options in the Pdepth menu allow the export of symbols for display in GM-SYS Profile and the creation of GM-SYS Profile models, as described below.

Export Solutions for Display in GM-SYS Profile

The *Export Solution for GM-SYS* menu option exports solutions from the current database into GM-SYS symbol (.SOL) files. It creates a symbol file for each selected line in the current database. It also creates a .GMS (symbol configuration) file for GM-SYS Profile model(s) to be made from the selected line(s) in the database.

To generate symbol files suitable for loading into GM-SYS models, make the solution database active and select **Export solutions for GM-SYS** from the *Pdepth* menu. This will generate a separate symbol file for each selected line in the solution database. The line name is appended to the base file name.



The dialog box titled "Export Depth Solutions for GM-SYS" contains the following fields and controls:

- File name root (+line #):** A text input field containing "dep_sol" with a browse button (...).
- File extension (sol):** A text input field containing "sol".
- Export template:** A text input field containing "mag_depth" with a browse button (...).
- Distance units:** A dropdown menu currently set to "kilometers".
- Buttons:** "OK", "Cancel", "Template", and "Symbols" are located at the bottom right.

The depths exported are those in the "Depth_sl" channel by default. If you want the clustered solutions exported, they need to be copied into the "Depth_sl" channel or you may make changes to the export template (the default template mag_depth.o0 can be found in "C:\Program Files\Geosoft\Desktop Applications\etc) to output the appropriate channel.

Creating GM-SYS Profile Models

The *Export Profiles to GM-SYS* menu option creates GM-SYS Profile models from all selected lines in a database in one pass. Up to 15 surfaces and observed gradient data from channels for can be included in the model.

To Generate GM-SYS Models from the Profile Database

1. Make sure the anomaly (not solution) database is selected.
2. From the *PDepth* menu, select **Export profiles to GM-SYS**.

The *New GM-SYS model from each selected line* dialog appears. This dialog will generate a separate GM-SYS model for each selected line in the database with the magnetic, topography, and gravity profiles.

New GM-SYS model from each selected line

Root name for models:

Magnetic channel:

Magnetic elevation channel:

OR constant magnetic elevation:

Gravity channel:

Gravity elevation channel:

OR constant gravity elevation:

Topography channel:

OR constant topography elevation:

Distance units:

Max Number of points in profile:

Horizons> Gradients> **Finish** Cancel

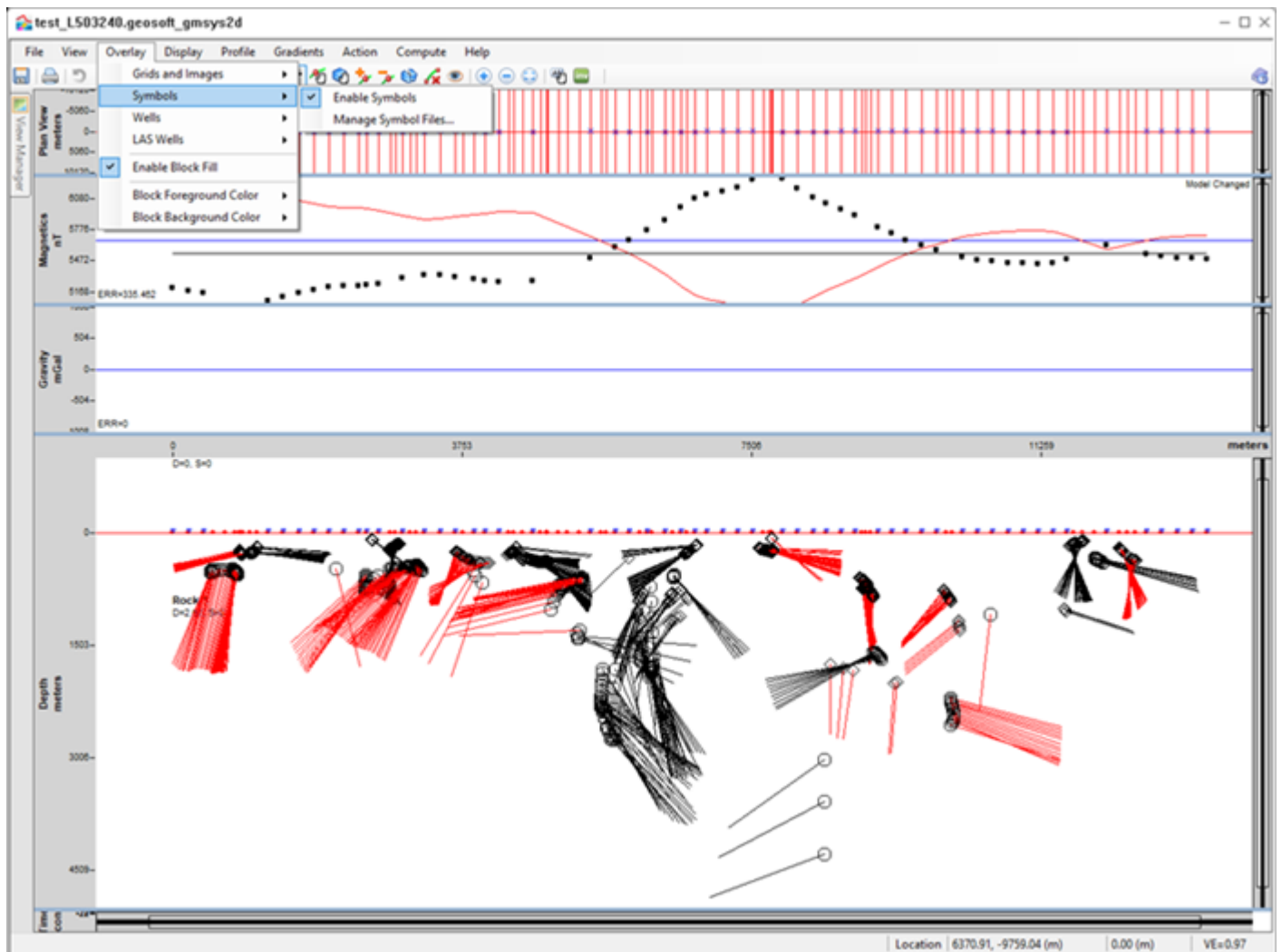
3. Specify the base *Root name for models*. You can also select the Magnetic and/or Gravity, Elevation and Topography channels, as well as selecting the Distance units.
4. Click **OK**.

GM-SYS Profile will not open automatically.

5. Open the imported model in **GM-SYS**.

You need to load the symbol file generated in the first step after opening the model in GM-SYS.

The image below shows an example of a GM-SYS model created using the above option, with the solutions displayed.



Solutions are displayed in GM-SYS via the *Symbols* option within the *Overlay* menu, as shown above.

Creating GM-SYS Profile Models and Exporting Solutions in One Pass

The *Export solutions to new GM-SYS model* menu option combines the export of solutions and the creation of new GM-SYS Profile models described above into a single operation. As described above, the symbols will still need to be displayed "manually" once the GM-SYS model is open.