S1 Overview

This Supplement provides a detailed introduction to the operational procedures of G&M3D 1.0, primarily covering the 3D modeling module and the potential field forward modeling module.

All software interface screenshots shown in this Supplement are from G&M3D 1.0, which was developed by the authors.

S2 Running G&M3D 1.0

G&M3D 1.0, developed based on the Qt framework (v6.9.1), has been released with all dependency libraries pre-packaged. Users can directly execute the **GM3D.exe** file in the "GM3D\build\Desktop_Qt_6_9_1_MinGW_64_bit-Release\release" directory to run the software and access the main interface shown in Fig. S1. For development purposes, we also provide the complete source code of G&M3D 1.0. Users who intend to perform independent development should download the open-source Qt framework (Index of /archive/online_installers/4.10) separately.

Developers should note that since the forward modeling algorithm employs FFT, the following dynamic link libraries must be copied to either the debug or release folder during debugging: libfftw3-3.dll, libfftwf3-3.dll, and libfftwl3-3.dll.

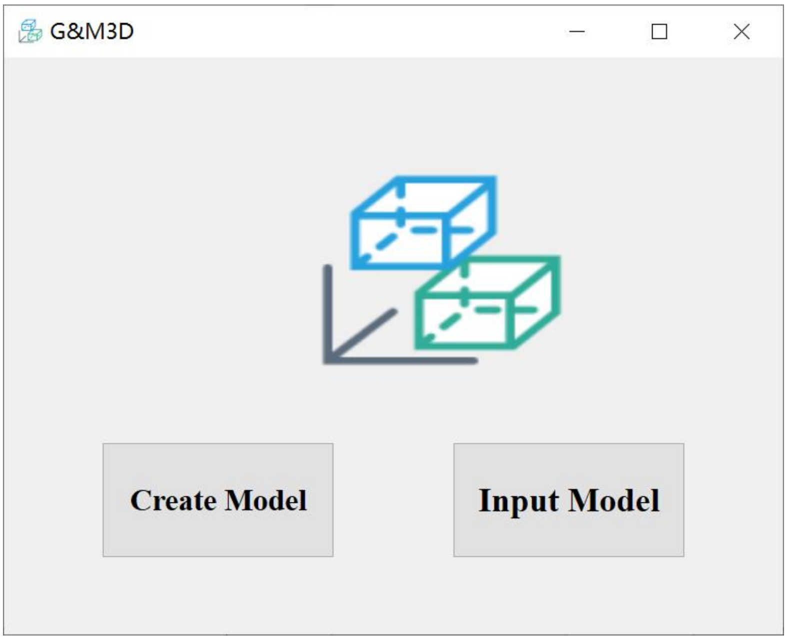

Fig. S1. Main interface of G&M3D 1.0.

S3 Creating a model

Click the "**Create Model**" button on the main interface (Fig. S1) to enter the 3D modeling module. If existing .bin files exported from G&M3D 1.0 are available, users may enter the 3D modeling module via the "**Input Model**" button to modify the source models.

After entering the modeling module, let's first introduce the layout of the 3D modeling module shown in Fig. S2. On the left side, the "**Body List**" showcases all established models with delete and modify functionality. The central region features a 3D model visualization, with the right side displaying orthogonal 2D cross-sectional views of the source models. The interface comprises two main sections at the top: "**Add Body**" and "**Operate**", which will be described in detail below. The

“**Add Body**” section centralizes all source model generation tools. In contrast, the “**Operate**” section combines the forward modeling module with auxiliary functions—including region configuration, file export, and cross-section visualization.

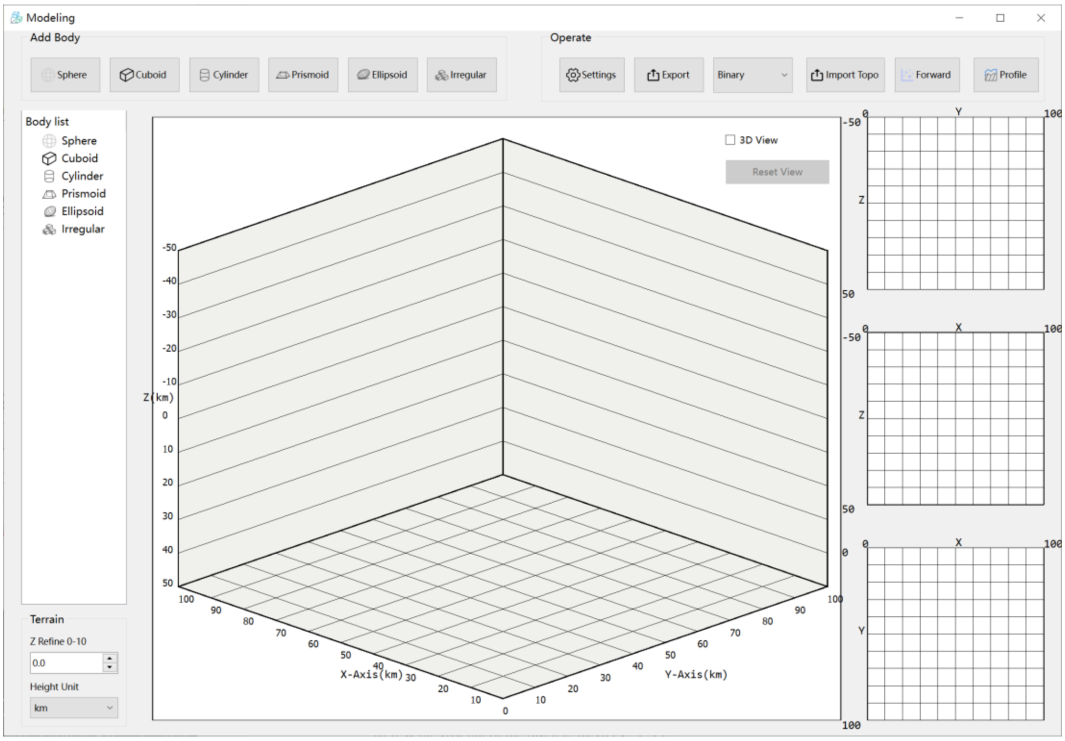

Fig. S2. 3D modeling module interface.

G&M3D 1.0 defaults to generating a 1 km × 1 km × 1 km grid within a 100 km × 100 km × 100 km source space to enable immediate modeling, though this configuration lacks customization. Typically, users should click the “**Setting**” button to define custom source space parameters that meet their specific project requirements.

In the Setting Field interface shown in Fig. S3, we need to enter the minimum values of spatial range in x, y, and z directions (Xmin, Ymin, Zmin) and maximum values (Xmax, Ymax, Zmax), as well as the spacing of the discrete mesh (X, Y, Z).

Three distance units are provided: meters (m), hectometers (hm), and kilometers (km). Users may select the appropriate unit according to their actual needs. After completing the settings, click “**ok**” to finalize the source model configuration and exit the Setting Field interface.

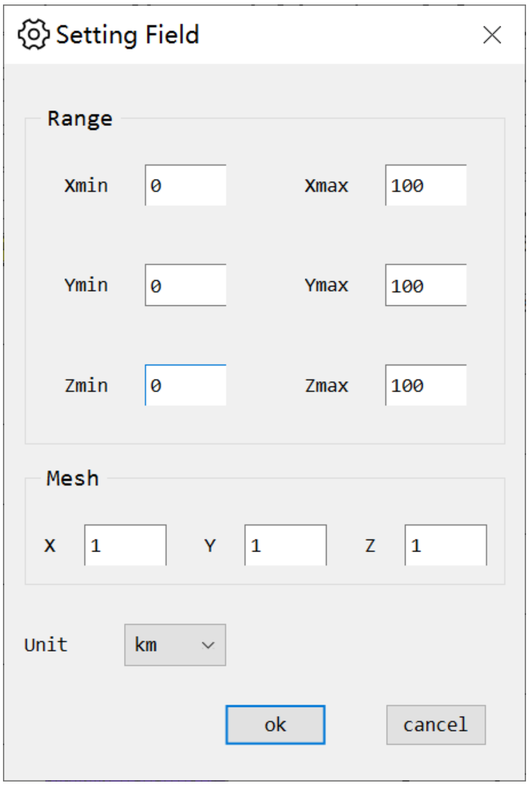

Fig. S3. Setting field interface.

After the source region is defined, users can import a topography file by clicking the "**Import Topo**" button. The first, second, and third columns of the topography file represent the *x*-, *y*-, and *z*-coordinates, respectively. The interface after importing topography is shown in Fig. S4. During model construction, anomalous bodies above the topographic surface are automatically removed, and users can only construct models below the topographic surface.

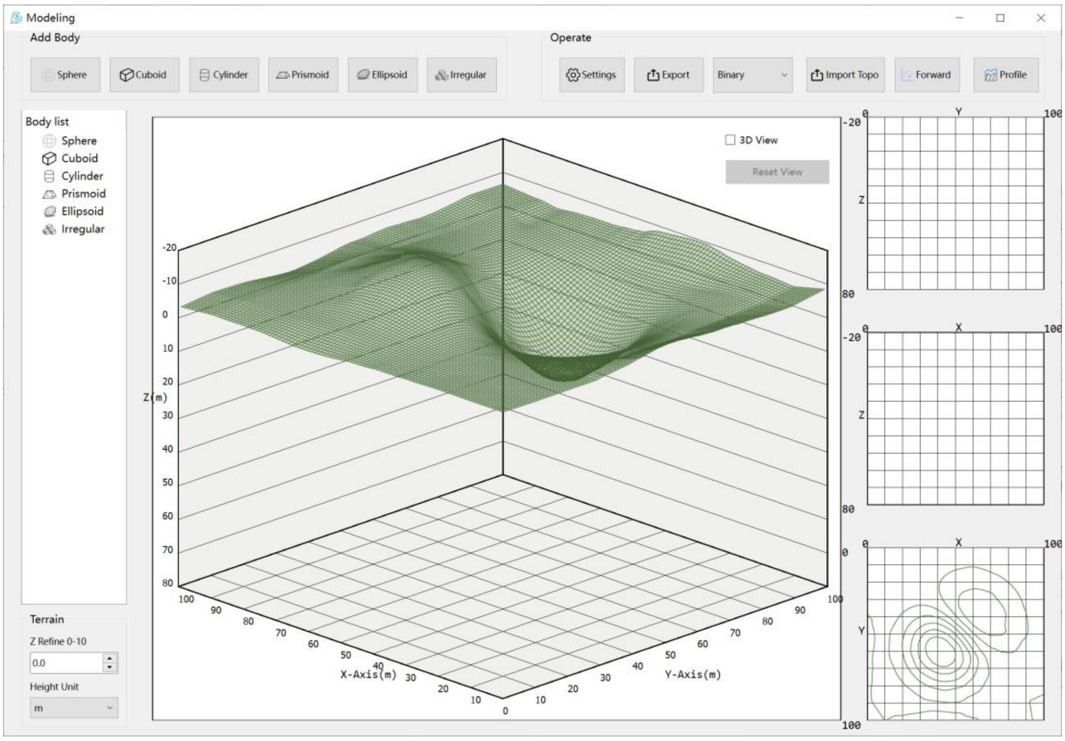

Fig. S4. Interface after importing topography.

Once the source region is configured, users can proceed to create their desired models. We broadly categorize models into two types: simple regular geometries and complex irregular structures. For regular models, we provide four primitive options: sphere, cuboid, cylinder, and prismoid. Users can click the corresponding button to enter the parameter configuration interface for each model type.

Firstly, the parameter configuration interface for the sphere model will be introduced, as shown in Fig. S5.

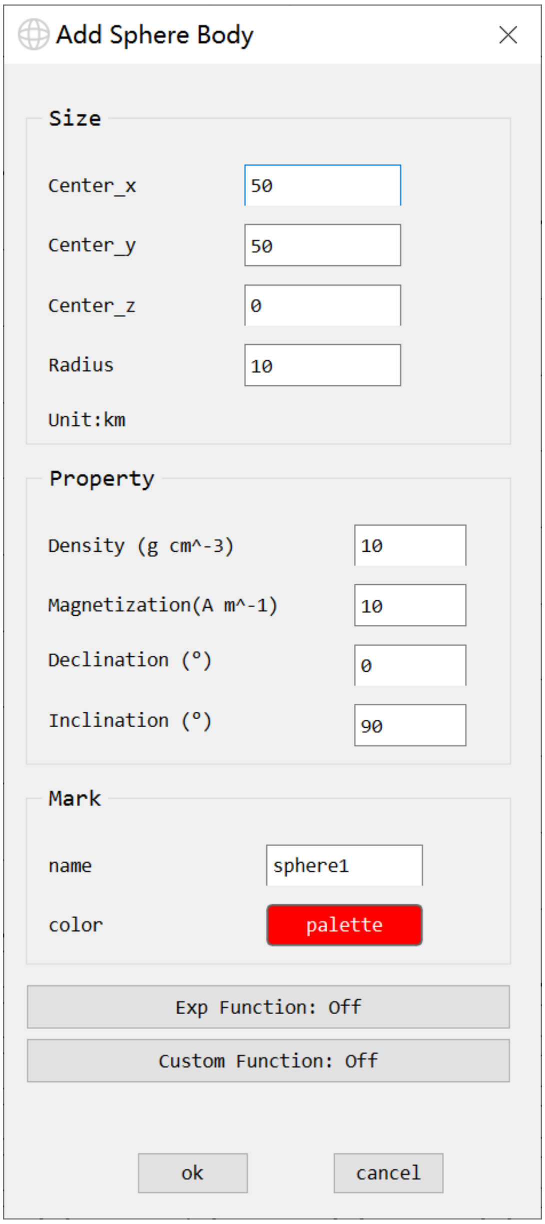

Fig. S5. Sphere model parameter setting interface.

To set up a sphere model, users should input the sphere's center coordinates in the "**Center_x**", "**Center_y**", and "**Center_z**" fields, specify the radius in the "**Radius**" field, enter the residual density in the "**Density**" field, set the magnetization intensity in the "**Magnetization**" field, and define the magnetic declination and inclination in the "**Declination**" and "**Inclination**" fields respectively. Additionally, in the "**Mark**" panel, users can assign a custom identifier to the sphere model in the "**Name**" field and select a distinguishing color via the "**Palette**" option to differentiate between various models.

After entering the parameters, click the "**ok**" button to return to the 3D modeling interface. At this time, you can see the schematic diagram of the created sphere model (Fig. S6) in the center area of the interface, and the model information in the model list on the left.

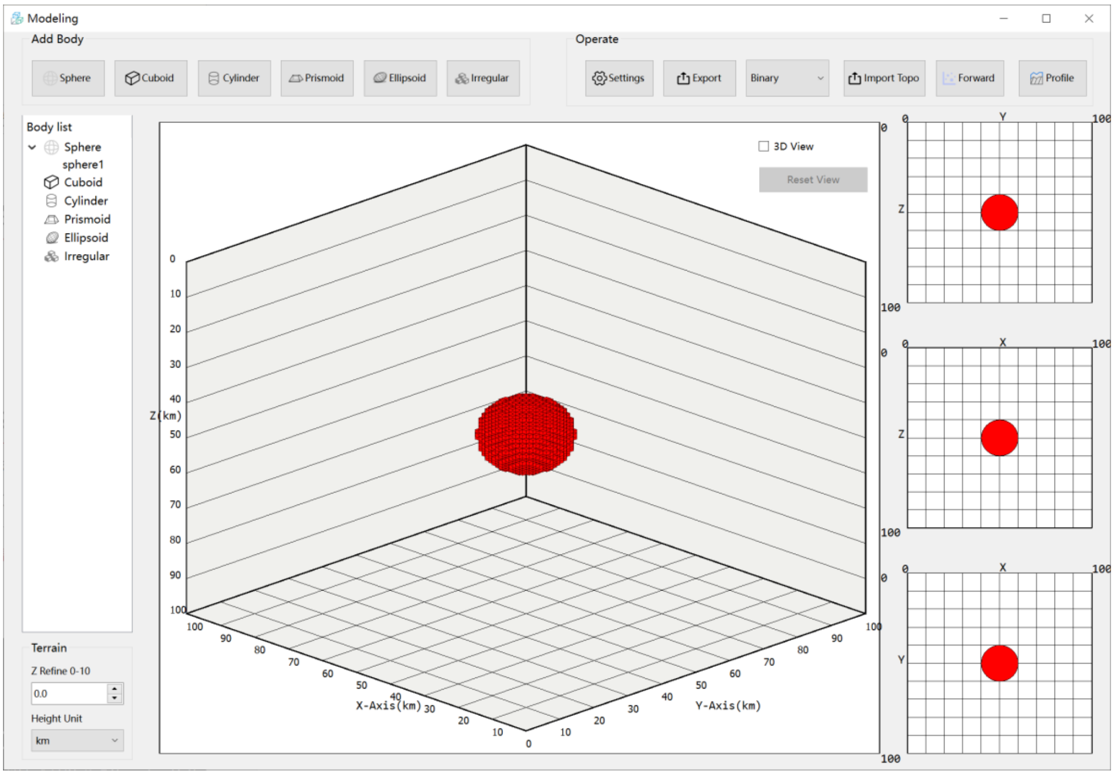

Fig. S6. Schematic diagram of the created sphere model.

Secondly, the parameter configuration interface for the cuboid model is presented in Fig. S7.

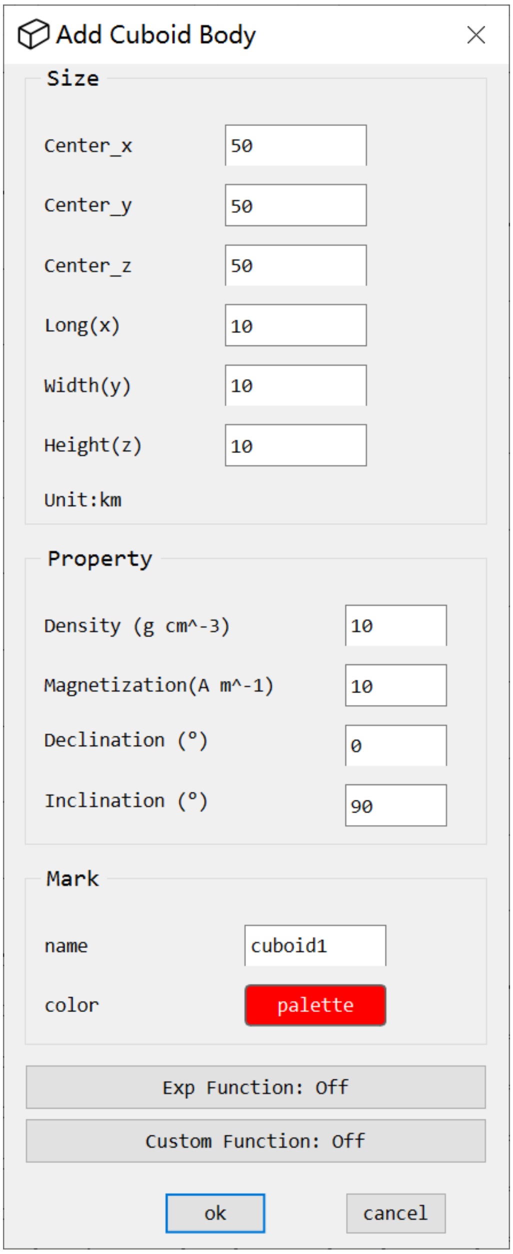

Fig. S7. Cuboid model parameter setting interface.

To create a cuboid model, users need to enter the geometric center coordinates in the "**Center_x**", "**Center_y**", and "**Center_z**" fields, and specify the dimensions along the X, Y, and Z axes in the "**Long**", "**Width**", and "**Height**" input fields, respectively. The configuration options in both the "**Property**" and "**Mark**" panels remain identical to those used for sphere models. Then click the "**ok**" button to finish creating the cuboid model. The completed source models are shown in Fig. S8.

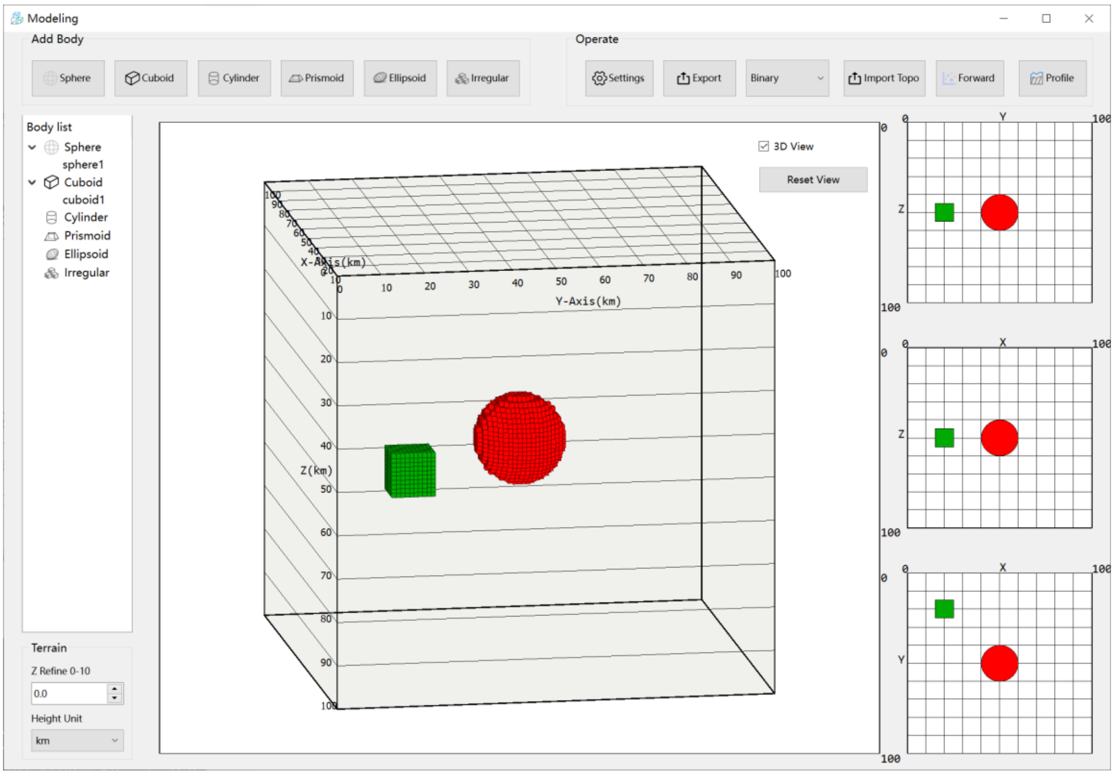

Fig. S8. Schematic diagram of the created cuboid model.

Subsequently, Fig. S9 displays the parameter configuration interface for the cylindrical model, presenting all essential input fields for model specification. To create a cylinder model, the "**Center_x**", "**Center_y**", and "**Center_z**" fields have the same physical meaning as in the sphere model. The "**Radius**" represents the cross-sectional radius of the cylinder. These four parameters are configured in the same way as in the sphere model, and users can follow the same input method used for the sphere model. The special parameters are as follows: users need to select the geometric extension direction from the "Direction" dropdown box and enter the extension length of the cylinder in the "**Length**" field. Then click the "**ok**" button to complete the creation of the cylinder model.

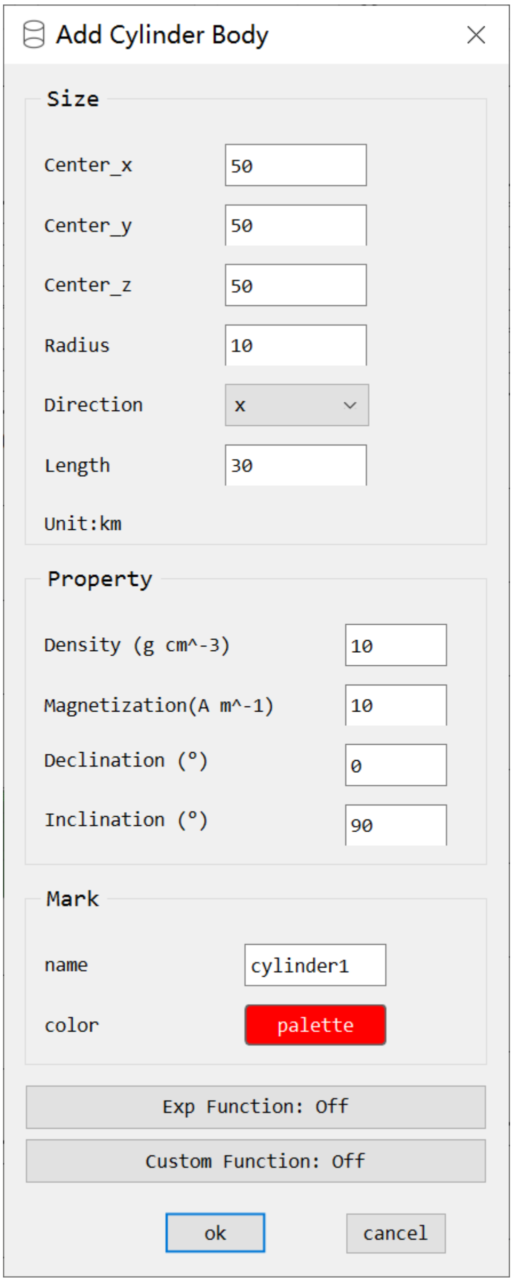

Fig. S9. Cylinder model parameter setting interface.

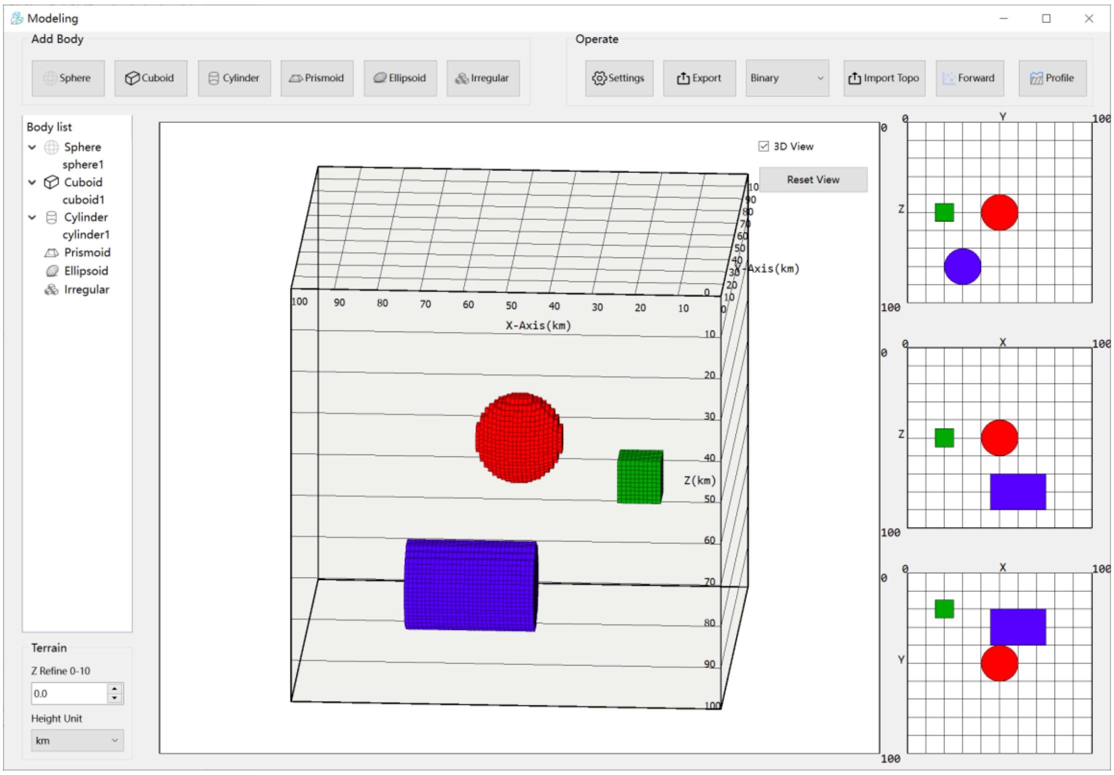

Fig. S10. Schematic diagram of the created cylinder model.

Next, the prismoid model serves to diversify the styles of regular geometric models, with its parameter configuration interface displayed in Fig. S11. To create a prismoid model, users must first determine the normal direction of two parallel planes through the "**Direction**" dropdown menu, then input the coordinate values for these planes in the "**Height1**" and "**Height2**" fields, ensuring "Height1" remains less than "Height2". Inputs labeled with "top" correspond to the coordinates of four edges on the "Height1" plane, while those labeled "bottom" relate to coordinates on the "Height2" plane. Taking the z-direction as an example, users need to enter: the x-coordinates of four edges parallel to the y-axis in the "**Top x1**", "**Top x2**", "**Bottom x1**", and "**Bottom x2**" fields; and the y-coordinates of four edges parallel to the x-axis in the "**Top y1**", "**Top y2**", "**Bottom y1**", and "**Bottom y2**" fields. Critical constraints require: topx1 < topx2, topy1 < topy2, bottomx1 < bottomx2, and bottomy1 < bottomy2. Then click the "**ok**" button to complete the creation of the prismoid model (Fig. S12).

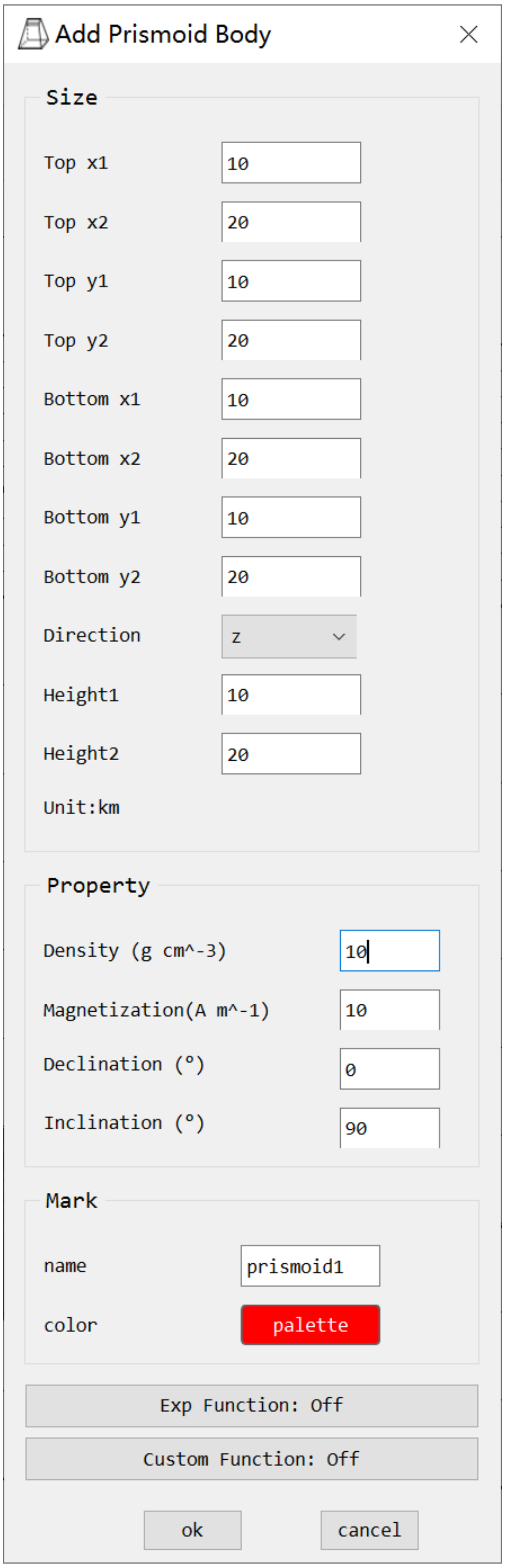

Fig. S11. Prismoid model parameter setting interface.

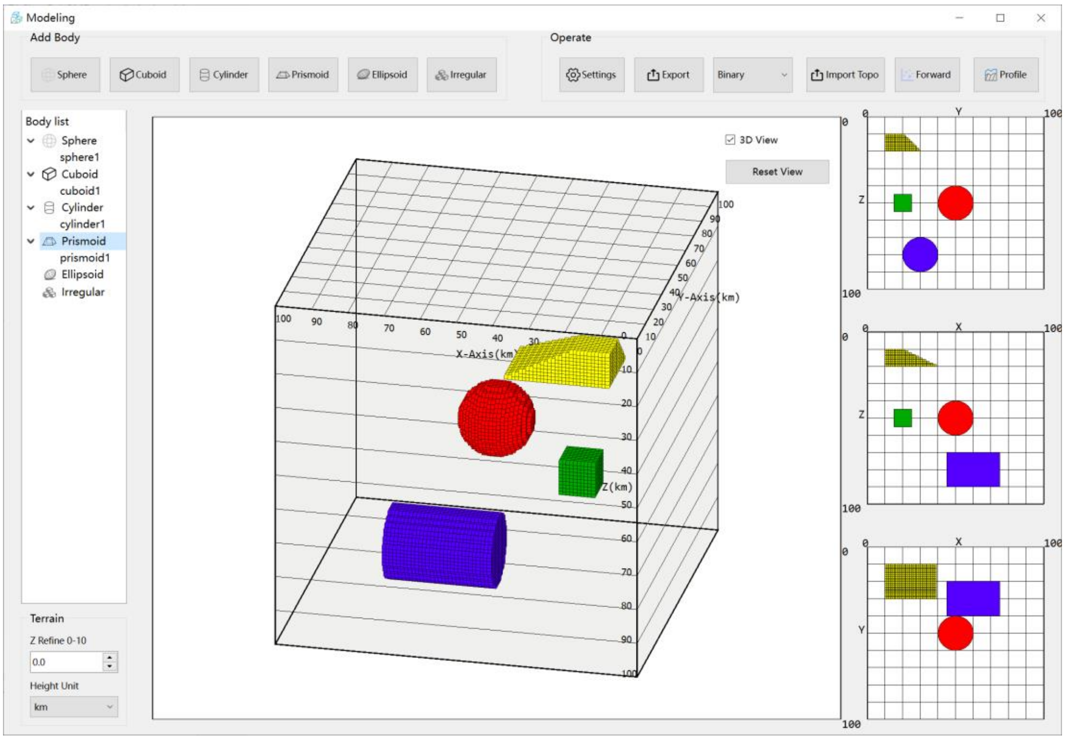

Fig. S12. Schematic diagram of the created prismoid model.

Finally, users can define a rotated ellipsoid, and the required parameters are shown in Fig. S13. The rotated ellipsoid is defined by the semi-axis lengths in three directions (**Rx**, **Ry**, and **Rz**), the coordinates of the ellipsoid center (**Center_x**, **Center_y**, and **Center_z**), and the rotation angles (**RotX**, **RotY**, and **RotZ**). After these parameters are specified, clicking "ok" generates the model, as shown in Fig. S14.

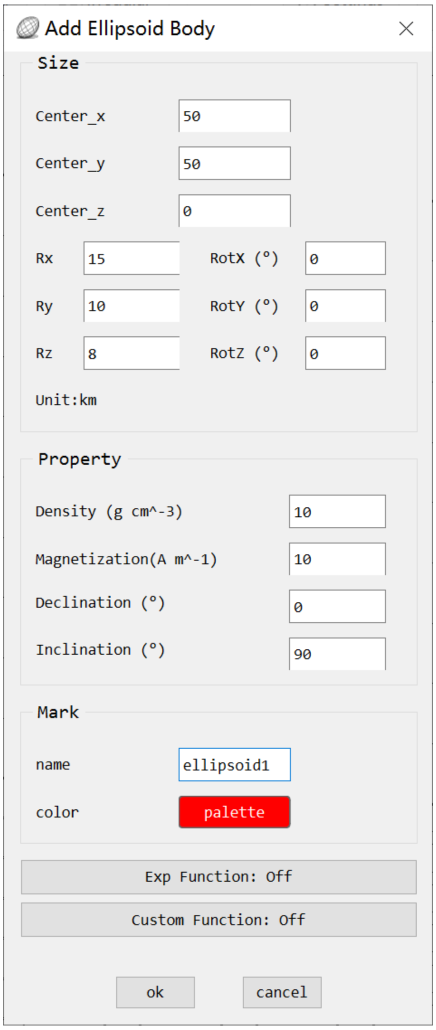

Fig. S13. Ellipsoid model parameter setting interface.

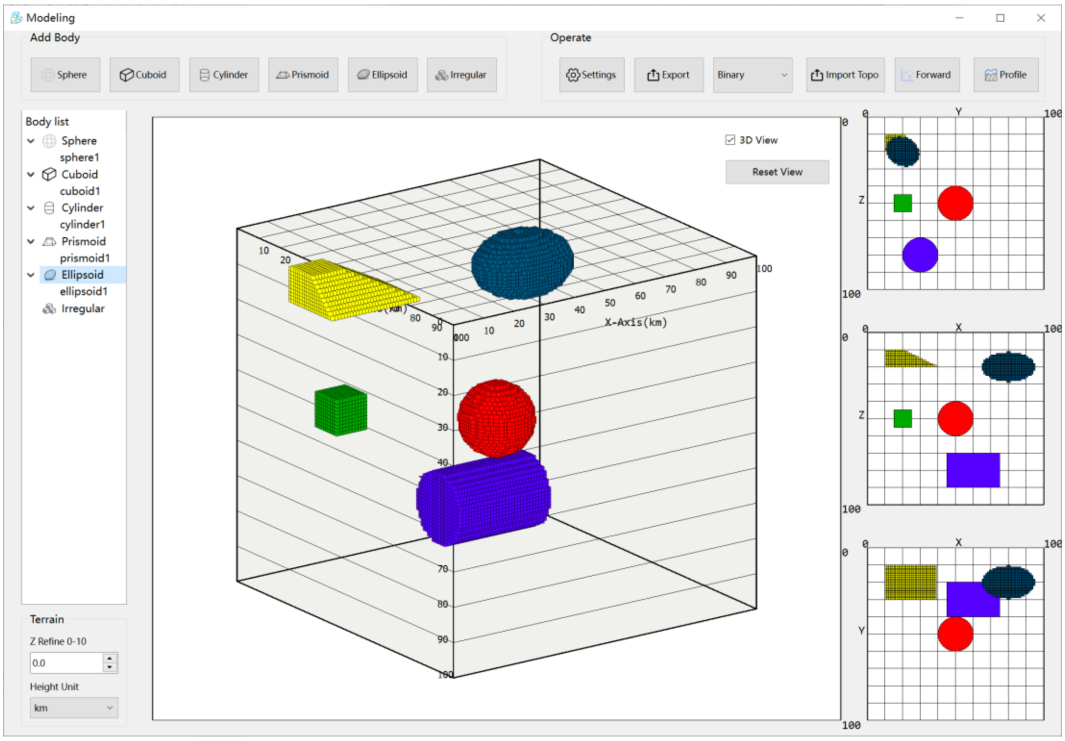

Fig. S14. Schematic diagram of the created ellipsoid model.

This concludes the tutorial on creating regular models. While model-specific parameters are configured exclusively in the "**Size**" panel, both the "**Property**" and "**Mark**" panels maintain uniform input configurations across all model types. For the same anomalous body, users can assign continuously varying physical-property parameters. The software provides a built-in definition for exponential distributions (Fig. S15). It also provides a customized definition button (Fig. S16), through which users can compile their own C++ scripts into functions to modify physical-property parameters at different locations.

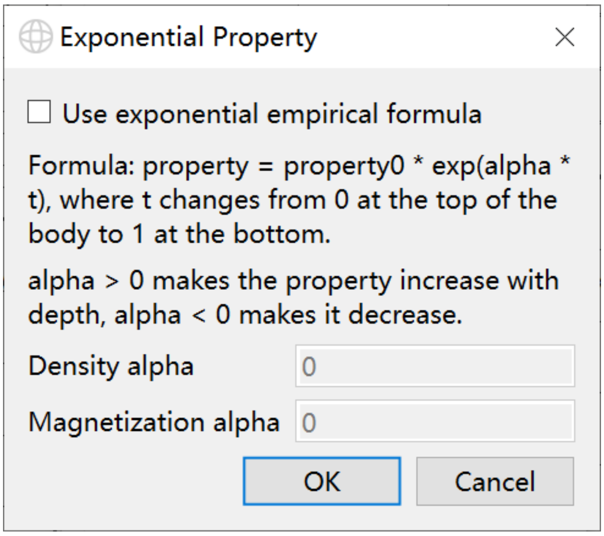

Fig. S15. Exponential function parameter-setting interface.

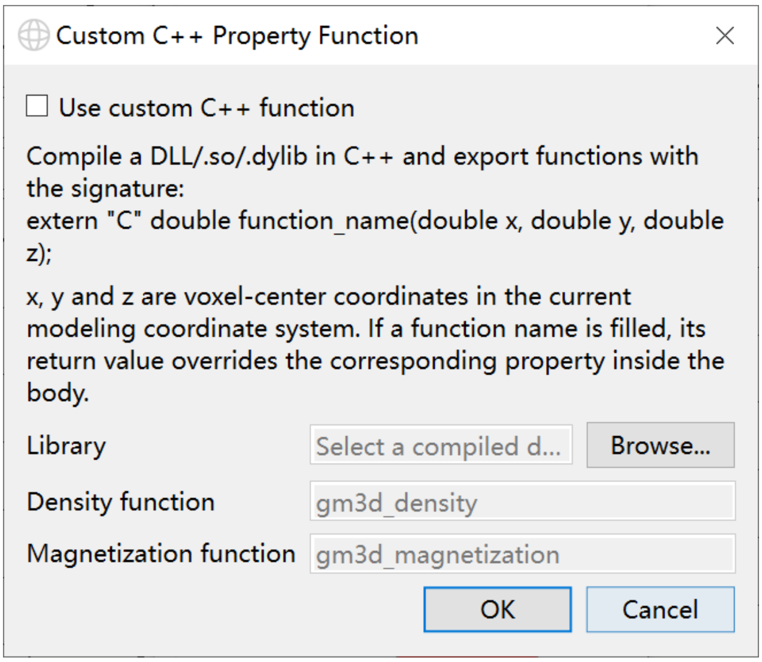

Fig. S16. User-defined function parameter-setting interface.

We provide a convenient tool for creating irregular models. Click the "**Irregular**" button to enter the "**Add Irregular Model**" tool (Fig. S17). Before creating an irregular model, we usually divide the model into layers in the X, Y, or Z directions. In this tool, we can create a layer of irregular anomalies by outlining the profile. Repeat until all layers of the irregular body are created. The user must select the layer-by-layer creation direction via the "**Profile**" dropdown menu and enter the target layer index in the "**Layer**" field. During irregular modeling, users can import an image using "**Background Mask (XY)**" as a semi-transparent mask to assist drawing.

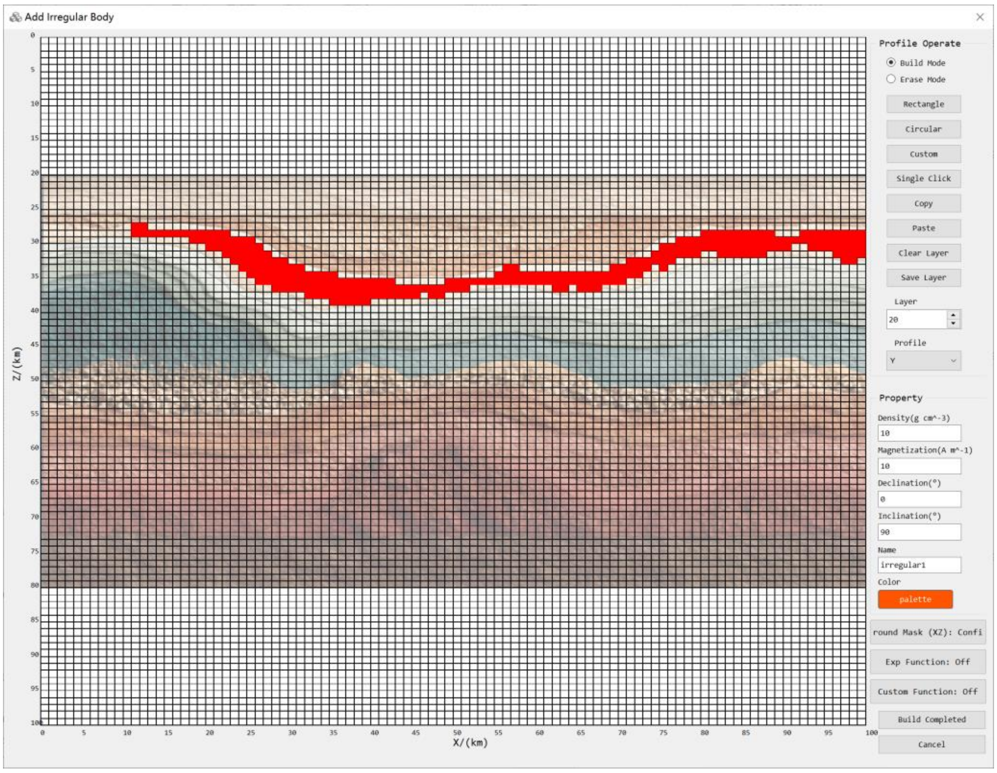

Fig. S17. Interface of the add irregular model tool.

To facilitate user operations, we provide two modeling modes: click "**Build Mode**" to create irregular models, and click "**Erase Mode**" to remove portions of existing modeled areas. We have configured four distinct modeling approaches: "**Rectangle**", "**Circular**", "**Custom**", and "**Single Click**". For rectangle mode, press and hold the left mouse button in the drawing area, drag from the top-left to the bottom-right corner, then release to create a rectangle. In circular mode, click and drag within the drawing area to generate a circle. The custom mode allows the creation of closed shapes with any geometry to form models. In contrast, single-click mode enables instant model generation by simply clicking and releasing in the drawing area. The "**Copy**" button duplicates existing models on the current layer, "**Paste**" inserts them onto the current layer, and "**Clear Layer**" removes all models from the current layer.

The parameters in the "**Property**" panel remain consistent with those of regular models, with the distinction that they only control the current layer. After completing both the model drawing and parameter configuration for the current layer, click "**Save Layer**" to automatically proceed to the next layer. Layer navigation across different levels can be achieved through the "**Layer**" input box. Users must explicitly click "**Save Layer**" after configuring each layer to preserve settings. When all target layers have been created, click the "**Build Completed**" button to save and return to the 3D modeling module, where the irregular model created by users will be displayed (Fig. S18).

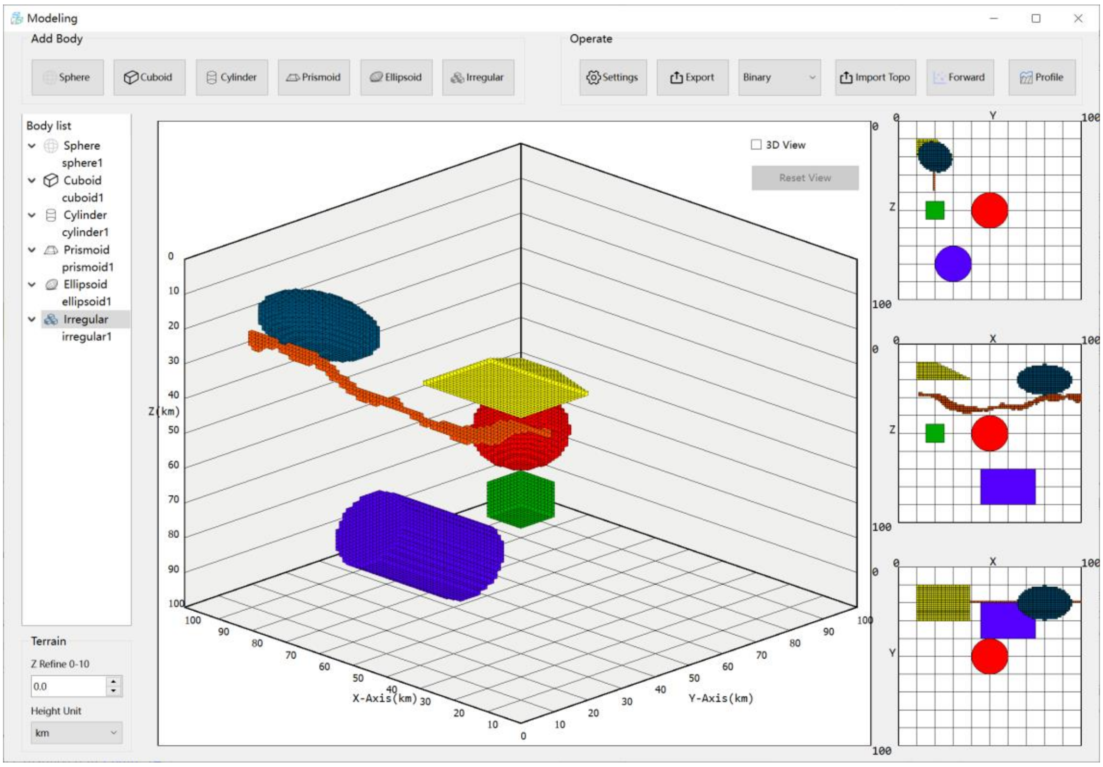

Fig. S18. Schematic diagram of the created irregular model.

To modify an existing model, right-click the target model in the "**Model List**" panel within the Model Module interface and select "**Modify**". Regardless of whether modifications are made, users must click "**ok**" or "**Build Completed**" to confirm and save the model. The "**Operate**" panel offers two model data export options: clicking the "**Model**" button generates a .bin format for native processing within G&M3D 1.0, while clicking the "**Data**" button exports a .txt file for viewing and use in external software.

For cross-sectional analysis, users may click the "**Profile**" button to access the sectional view, with the corresponding interface displayed in Fig. S19.

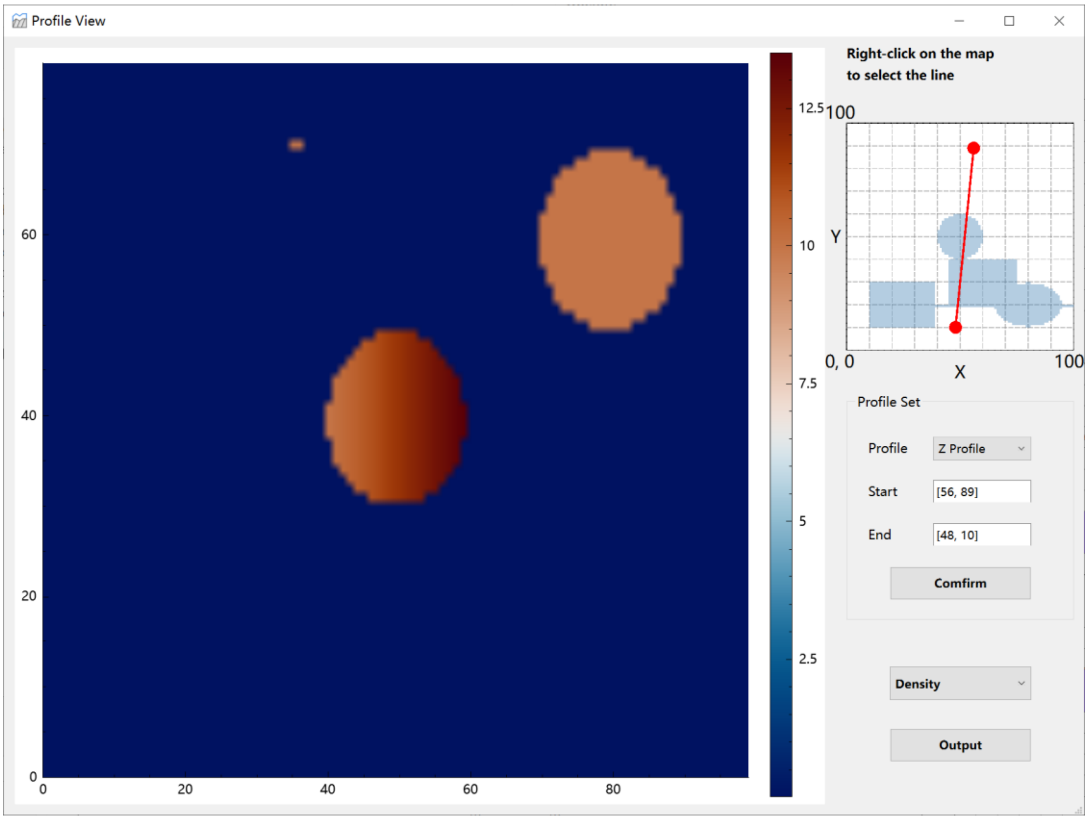

Fig. S19. Interface for generating cross-sectional views of the source model.

To generate a cross-sectional view of the source model, users should first select the normal direction of the section plane from the "**Profile**" dropdown menu, then sequentially click two endpoints in the coordinate area located at the upper-right corner. Clicking the "**Confirm**" button will generate the profile diagram.

Upon completing the source model creation, users may click the "**Forward**" button to enter the forward calculation module.

S4 Performing forward modeling

Both the "**Create Model**" and "**Input Model**" options from the main interface will direct users to the Modeling interface. By clicking the "**Forward**" button, users can enter the Forward Modeling interface (Fig. S20) to initiate forward calculations.

When entering the Forward Modeling module, the left section serves as the result visualization area. At the same time, the upper-right corner contains the "Model List" panel, which remains consistent with the Modeling interface. The "**Forwarding**" button located at the lower-right corner allows configuration of forward modeling parameters for both the gravity field and its gradient, as well as the magnetic field and its gradient. Clicking the "**Forwarding**" button will commence either the gravity forward calculation or the magnetic forward calculation.

Fig. S20. Forward modeling module interface.

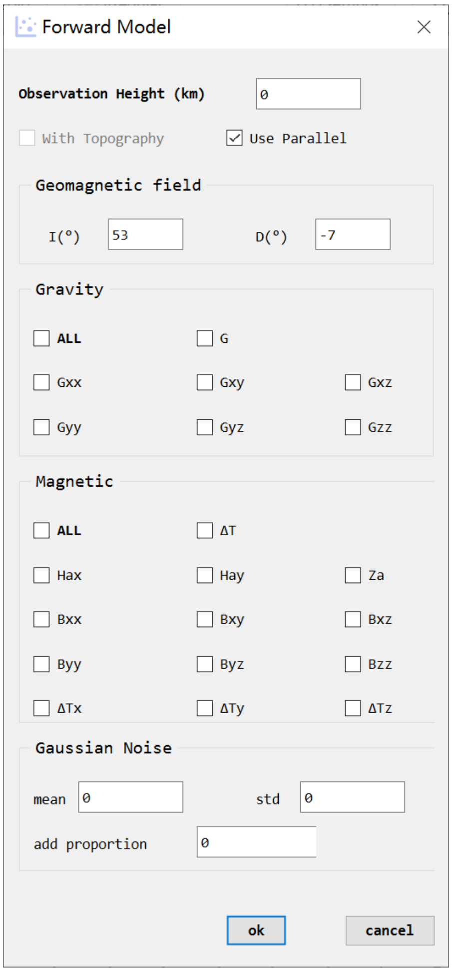

Fig. S21. Forward modeling parameter input interface.

Users should input the observation altitude in the "**Observation Height**" field and specify the geomagnetic field's declination and inclination angles in the "**Geomagnetic Field**" section. Within the "**Gravity**" panel, select the required gravity field components and their derivatives for computation. In contrast, the "**Magnetic**" panel provides options for choosing the desired magnetic field components and related parameters. The software supports forward calculations with and without topography, and users can also choose whether to enable parallel computation.

After completing all forward modeling parameter settings, click the "**ok**" button to initiate the calculation. Upon completion, a timing notification window will appear, as shown in Fig. S22, and the forward modeling results will be displayed in Fig. S23.

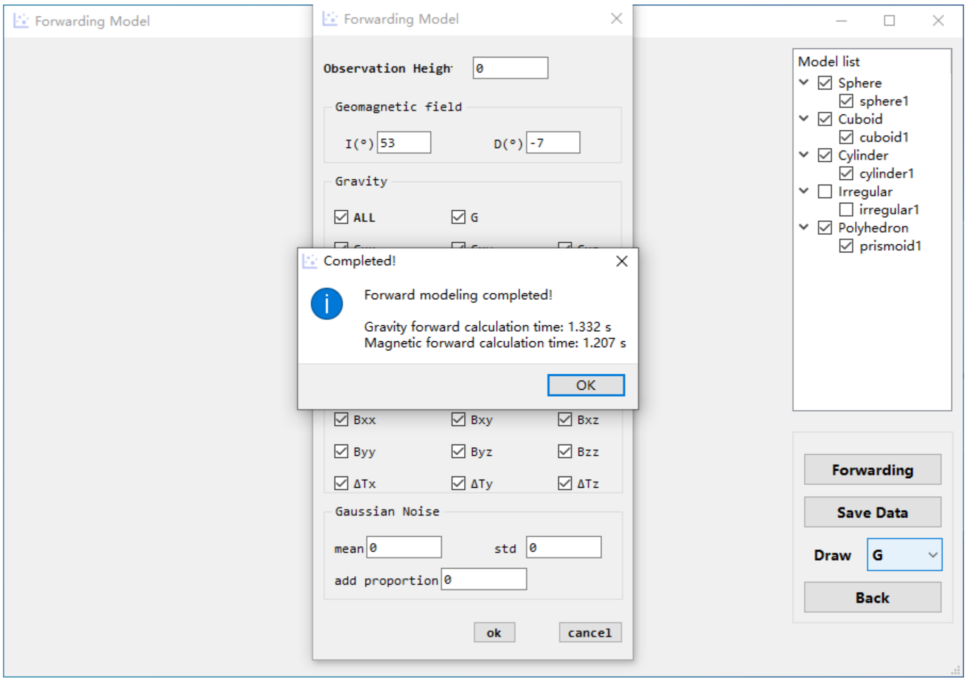

Fig. S22. Forward modeling timer pop-up diagram.

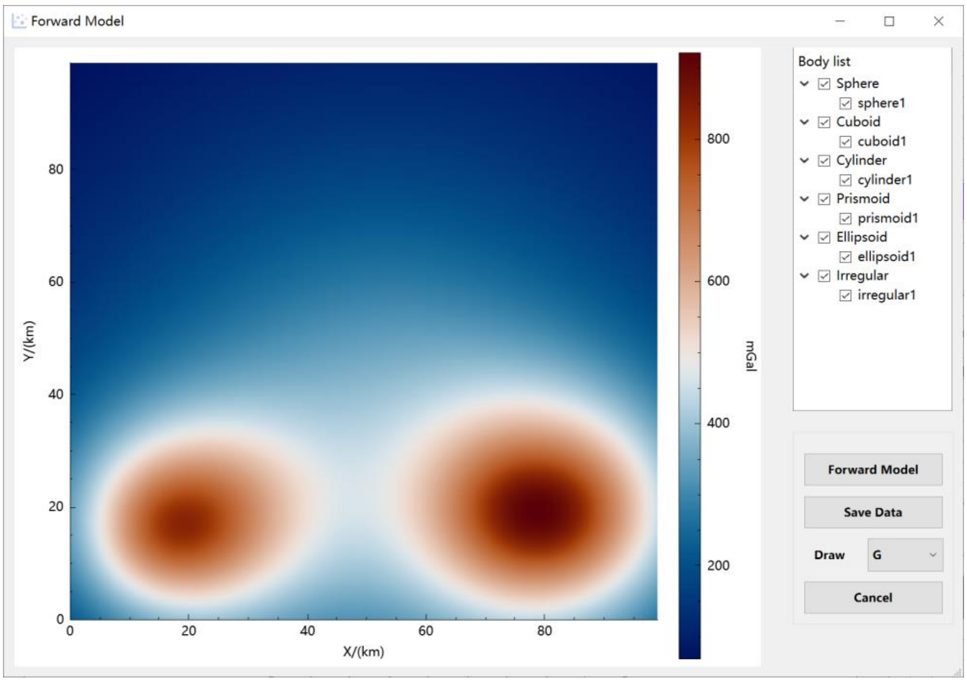

Fig. S23. Visualization interface for forward modeling results.

Users can select the field quantities for visualization through the "**Draw**" dropdown menu. G&M3D 1.0 supports the following measurable fields: gravity field (G), gravity gradient components (Gxx, Gxy, Gxz, Gyy, Gyz, Gzz), magnetic field (T), horizontal magnetic components (Hax, Hay), vertical magnetic intensity (Za), and magnetic gradient components (Bxx, Bxy, Bxz, Byy, Byz, Bzz, Tx, Ty, Tz).

For Gaussian noise configuration, enter the relevant parameters in the dedicated panel: input the mean value in the "**mean**" field, define the standard deviation in "**std**", and control the noise application scope through "**add proportion**" (enter 0 for no noise, a decimal between 0-1 for partial coverage, or 1 for full noise implementation). The computation results with Gaussian noise added are shown in Fig. S24.

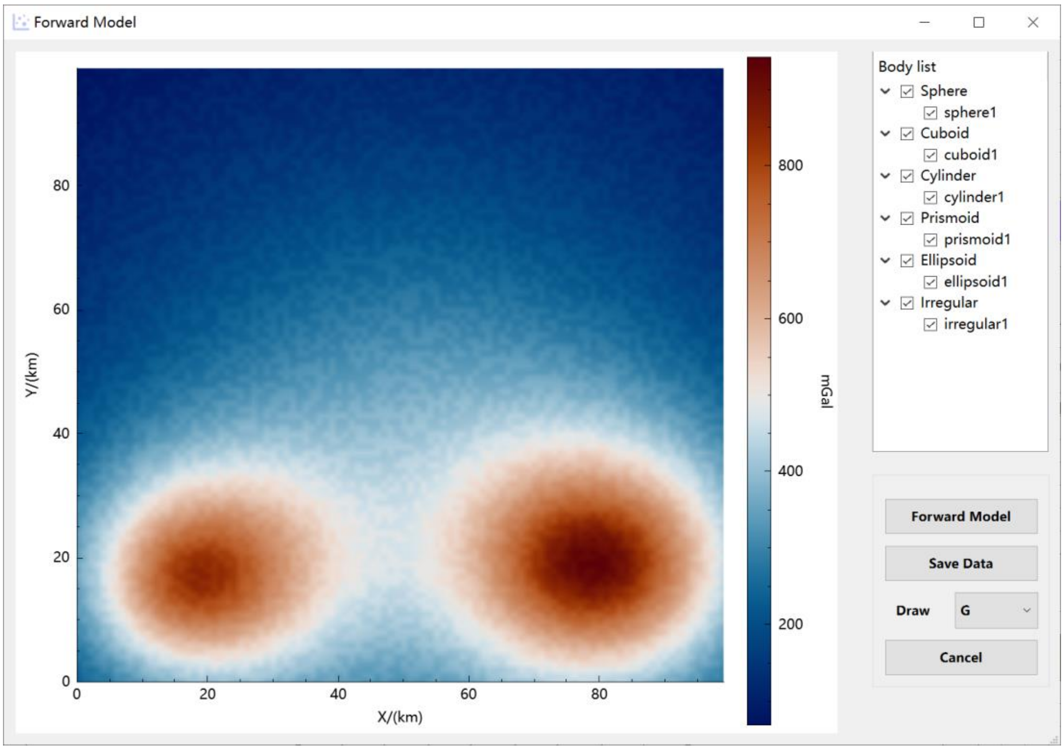

Fig. S24. Forward modeling result after adding Gaussian noise.

After the calculation is over, click the "**Data**" button to view the performance data and export it to a file.

This Supplement describes the primary workflow of G&M3D 1.0. Further information is available in the main manuscript and the associated code repository.